USER' S MANUAL

DeepXcav software program (Version 2011)

(ParatiePlus within Italy)

Version 1.0

Issued: 20-May-2011

Deep Excavation LLC

www.deepexcavation.com

Table of Contents

CHAPTER1: INTRODUCTION TO DeepXcav 5
1.1 About DeepXcav (Deep Excavation Engineering Program) 6
1.2 Software Compatibility & Installation 7
1.3 Support & Technical Assistance 7
1.4 End User License Agreement 7
1.5 What is new! 10
1.6 Activating the software 10
1.6.1 Activating the stand alone license 10
1.6.2 Activating the network license 12
CHAPTER 2: USING DeepXcav 15
2.1 Deep Excavations Background 16
2.1.1. Background 16
2.1.2. RETAINING SYSTEMS FOR DEEP EXCAVATIONS 17
2.1.2. A. Soldier Pile and Lagging Walls 17
2.1.2. B. Sheet Pile Walls 19
2.1.2. C. Secant Pile Walls 20
2.1.2. D. Soil Mix Walls 21
2.1.2. E. Diaphragm Walls (Slurry walls US) 22
2.1.3 SUPPORT SYSTEMS FOR DEEP EXCAVATION: 23
2.1.3. A. Tiebacks/Rock Anchors 23
2.1.3. B. Cross-lot/Internal Bracing 25
2.1.3. C. Top/Down Construction 27
2.2 Using DeepXcav 28
2.3.1 DeepXcav Toolbar Functions 31
2.3.2 Wall List, Design Section List, and Project Tree View 35
2.4 General menu 36
2.5 Properties menu 48
2.6 Model menu 59
2.7 Load/Support menu 67

2.8 Seismic menu 71
2.9 Analysis menu 75
2.10 Slope Stability menu 88
2.11 Stability+ menu 94
2.12 Design menu 97
2.13 Results menu 100
2.14 Report menu 102
2.15 View menu 105
2.16 Optimize menu 106
2.17 Help menu 109
CHAPTER 3: DATA ENTRY 110
3.1 Data entry: General 111
3.2 Design codes 111
3.3 Data entry: Project information 118
3.4 Data entry: Soil Data 119
3.5 Data entry: Soil Layers 127
3.6 Data entry: Water 128
3.7 Data entry: Wall data 129
3.7.1 Data entry: Wall sections 130
3.7.2 Wall type: Soldier piles 135
3.7.3 Wall type: Sheet piles 142
3.7.4 Wall type: Secant - tangent piles 145
3.7.5 Wall type: Diaphragm walls (slurry walls US) 150
3.7.6 Wall type: Soldier pile and tremied concrete walls 154
3.7.7 Wall type: Custom 155
3.8 DATA ENTRY 157
3.8.1 Data entry: Tiebacks 157
3.8.2 Data entry: Tieback sections 158
3.8.3 Data entry: Helical anchor sections 163
3.9 Data entry: Strut sections 165
3.10 Data entry: Slab sections and slab supports 167

3.11 Data entry: Fixed supports 170
3.12 Data entry: Springs 170
3.13 Data entry: Slope Stability 172
3.13.1 Slope Stability Analysis 172
3.13.2 Soil nailing 184
3.14 Data entry: Wale beams 190
CHAPTER 4: MODIFYING MODELS & VIEWING RESULTS 199
4.1 Modifying surface elevations 200
4.2 Adding supports graphically 204
4.3 Adding Surcharges Graphically 207
4.4 Viewing Results on Main Form 209
4.5 Report Options (Printed Reports) 215
4.6 Footing Loads (3-Dimensional Loads) 217
4.7 Buildings & Building Wizard 219

CHAPTER1: INTRODUCTION TO DeepXcav

1.1 About DeepXcav (Deep Excavation Engineering Program)

DeepXcav is a user friendly robust software program that evaluates and aids the design of deep excavations. The design of deep excavations can be a very complicated matter. The designer has to content with many unknowns and factors that influence the behavior of the excavation. Typically, there are two systems in excavations that must be designed: A) the Earth Retention System that contains the earth i.e. the support wall (sheet pile, diaphragm wall, etc.), and b) the Support System (i.e. the internal or external bracing such as rakers, struts, or tiebacks) that supports the earth retention system.

Performing detailed calculations for both systems can be a very time consuming process, especially when parameters have to be changed. In addition, many current software programs do not offer an integrated platform of structural and geotechnical analyses required to design deep excavations. As a result, the designer is forced to use numerous software programs to analyze the excavation and the structural system separately. With the exception of finite element analyses, there are very few theoretical solutions for calculating lateral soil pressures from complex surface profiles. Furthermore, the designer has to save under different filenames the different stages of the same excavation. As a result, the whole process can become unnecessarily complicated and time consuming. DeepXcav addresses most of these issues and provides an integrated structural and geotechnical platform for designing deep excavations.

The current version of DeepXcav offers both traditional methods of analyses and discrete soil spring solutions. While it is generally accepted that traditional methods of analysis have obvious limitations in predicting real behavior accurately, they are important for framing the problem and providing a back-check for more rigorous finite element methods. Soil spring solutions can give a better approximation to real wall behavior. The beauty of DeepXcav is that it can perform both traditional and spring analyses so efficiently that it leaves the designer with ample time to perform more time consuming finite element analysis.

1.2 Software Compatibility & Installation

DeepXcav is compatible with Windows (OS) XP, Vista and 7. A minimum of 380 Mb must be available on your hard disk.

1.3 Support & Technical Assistance

Support and technical assistance for DeepXcav is offered through our web site at:

www.deepexcavation.com

1.4 End User License Agreement

"TERMS OF USE / LICENSE AGREEMENT"

This legal document is an agreement between you (the end user) and Deep Excavation. BY CONTINUING WITH/OPENING/DOWNLOADING THIS SOFTWARE PROGRAM, YOU ARE AGREEING TO BECOME BOUND BY THE TERMS OF THIS AGREEMENT, WHICH INCLUDES THE SOFTWARE LICENSE, SOFTWARE DISCLAIMER OF WARRANTY, AND HARDWARE LIMITED WARRANTY "collectively the "Agreement".

This agreement constitutes the complete agreement between you and Deep Excavation. IF YOU DO NOT AGREE TO THE TERMS OF THIS AGREEMENT, DO NOT CONTINUE WITH THIS SOFTWARE PROGRAM. Promptly return or delete the software program (cd and jewel case) and other items that are part of this product to Deep Excavation, for a complete refund if a purchasing fee was charged.

DEEP EXCAVATION ENGINEERING PROGRAM,"SOFTWARE LICENSE", APPLICABLE TO ALL VERSIONS

THE LICENCE APPLIES TO PURCHASHED AND FREE OF PURCHASE VERSIONS OF THE DEEP EXCAVATION SOFTWARE.

LICENSE. In consideration of payment of the LICENSE fee, which is a part of the price you paid for this product, Deep Excavation, as Licensor grants to you, the LICENSEE, a nonexclusive right to use and display this copy of Deep Excavation Engineering Program, Software (hereinafter referred to as "SOFTWARE" on a single COMPUTER (i.e.,

with a single CPU) at a single location. Any "networking", namely operating this program on a "network" is strictly forbidden. You as a Licensee are strictly forbidden to operate, utilize, transfer, distribute, connect, network, link to, attach, or operate in any manner this software on the internet, worldwide web, via email, any website, networking, any multimedia device, electronic or otherwise or any form of electronic media whatsoever. This includes but is not limited to the written materials, results, output, or resulting answers and/or printed matter without the prior written consent of Deep Excavation. Deep Excavation reserves all rights not expressly granted to LICENSEE.

SOFTWARE OWNERSHIP. As the LICENSEE, you own the magnetic or other physical media on which the SOFTWARE is originally or subsequently recorded or fixed, however, Deep Excavation retains title and ownership of the SOFTWARE recorded on the original disk copy(ies) and any subsequent copies of the SOFTWARE, regardless of the form or media in or on which the original and other copies may exist. This License is not a sale of the original SOFTWARE or any copy thereof.

COPY AND/OR MODIFY RESTRICTIONS. All Licensed Products are copyrighted and may not be further copied, without the prior written approval of Deep Excavation except that You may make one copy for backup purposes provided You reproduce and include the complete copyright notice, disclaimer, etc., on the backup copy. Any unauthorized copying is in violation of this Agreement and also a violation of the United States Copyright law. You may not use, transfer, modify, copy of otherwise reproduce the License Product, or any part of it, except as expressly permitted in this End User License Agreement.

USE RESTRICTIONS. As the LICENSEE, you may physically transfer the SOFTWARE from one computer to another provided that the SOFTWARE is used on only one computer at a time. You may not electronically transfer the SOFTWARE from one computer to another over a network. You may not distribute copies of the SOFTWARE or accompanying written materials to others. You may not operate, utilize, transfer, distribute, connect, network, link to, attach, or operate in any manner this software on the internet, worldwide web, via email, any website, networking, any multimedia device, electronic or otherwise or any form of electronic media whatsoever. You may not modify, adapt, translate, reverse engineer, decompile, disassemble, or create derivative works based on the SOFTWARE. In addition, you may not modify, adapt, translate, or create derivative works based on the written materials, results, output, or resulting answers and/or printed matter without the prior written consent of Deep Excavation.

RESTRICTIONS AGAINST TRANSFER. This SOFTWARE is licensed only to you, the LICENSEE, and may not be transferred to anyone without the prior written consent of DEEP EXCAVATION. Any authorized transferee of the SOFTWARE shall be bound by the terms and conditions of this Agreement. In no event may you transfer, assign, copy, rent, lease, sell, or dispose of the SOFTWARE in any manner on a temporary or permanent basis except as expressly provided herein.

TERM. This End User License Agreement is effective from the date of purchase by You or granting to you of the Licensed Product and shall remain in force until terminated. You may terminate this End User License Agreement at any time by destroying the Licensed Product together with any backup copy in any form made by You or received by You. In addition, your right to use the Licensed Product will terminate if You fail to comply with any of the terms or conditions of this End User License Agreement. Upon such termination You shall destroy the copies of the Licensed Product in your possession.

DISCLAIMER OF WARRANTY AND LIMITED WARRANTY

THE SOFTWARE AND ACCOMPANYING WRITTEN MATERIALS (INCLUDING RESTRICTIONS FOR USE) IF ANY, ARE PROVIDED "AS IS" WITHOUT WARRANTY OF ANY KIND. FURTHER, DEEP EXCAVATION DOES NOT WARRANT, GUARANTEE, OR MAKE ANY REPRESENTATIONS REGARDING THE USE, OR THE RESULTS OF THIS USE, OF THE SOFTWARE OR WRITTEN MATERIALS IN TERMS OF CORRECTNESS, ACCURACY, RELIABILITY, CURRENTNESS, OR OTHERWISE. THE ENTIRE RISK AS TO THE RESULTS AND PERFORMANCE OF THE SOFTWARE IS ASSUMED BY YOU

.

Deep Excavation warrants to the original LICENSEE (a) the disk(s) on which the SOFTWARE is recorded to be free from defects in materials and workmanship under normal use and service for a period of sixty (60) days from the date of delivery as evidenced by a copy of the receipt. In addition, Deep Excavation hereby limits the duration of any implied warranty(ies) on the disk or such hardware to the respective period stated above.

Deep Excavation's entire liability and your exclusive remedy as to the disk(s) or hardware shall be, at Deep Excavation's option, either (1) return of the purchase price or (2) replacement of the disk or hardware that does not meet Deep Excavation's Limited Warranty and which is returned to Deep Excavation with a copy of the receipt. If failure of the disk or hardware has resulted from accident, abuse or misapplication, Deep Excavation shall have no responsibility to replace the disk or hardware or refund the purchase price. Any replacement disk or hardware will be warranted for the remainder of the original warranty period or thirty (30) days, whichever is longer.

THE ABOVE ARE THE ONLY WARRANTIES OF ANY KIND, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE IMPLIED WARRANTIES OR MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE THAT ARE MADE BY DEEP EXCAVATION ON THIS PRODUCT. NO ORAL OR WRITTEN INFORMATION OR ADVICE GIVEN BY DEEP EXCAVATION, ITS DEALERS, DISTRIBUTORS, AGENTS, OR EMPLOYEES SHALL CREATE A WARRANTY OR IN ANY WAY INCREASE THE SCOPE OF THIS WARRANTY, AND YOU MAY NOT RELY ON ANY SUCH INFORMATION OR ADVICE.

NEITHER DEEP EXCAVATION NOR ANYONE ELSE WHO HAS BEEN INVOLVED IN THE CREATION, PRODUCTION, OR DELIVERY OF THIS PRODUCT SHALL BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL, OR INCIDENTAL DAMAGES (INCLUDING DAMAGES FOR LOSS OF BUSINESS PROFITS, BUSINESS INTERRUPTION, LOSS OF BUSINESS INFORMATION, AND THE LIKE) ARISING OUT OF THE USE OF OR INABILITY TO USE SUCH PRODUCT EVEN IF DEEP

EXCAVATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES. IN ALL CASES A LICENCED PROFESSIONAL ENGINEER SHALL APPROVE AND STAMP ANY RESULTS BY D.E.E.P. AND THAT ENGINEER IS ULTIMATELY RESPONSIBLE FOR ANY CONSEQUENCES OR MISUSE OF THE SOFTWARE.

This Disclaimer of Warranty and Limited Warranty is governed by the laws of the State of New York. Should you have any questions regarding this agreement please email:

Deep Excavation,

deepexcavation@deepexcavation.com

1.5 What is new!

The latest version of DeepXcav 2011 comes with many exciting new features that include:

i. Walers – wall bracing.
ii. Slope stability and soil nailing analysis.
iii. Custom layer mode.
iv. Multiple wall beams.

1.6 Activating the software

1.6.1 Activating the stand alone license

In order to activate the license, the following steps are required;

1) Download and install the software.
2) Keep the SHIFT key pressed (or CAPS locked) and start DeepXcav 2011
3) The activation window should appear (Figure 1.6.1).
4) E-mail us the SITE and MID codes that appear in this window (see Figure 1.6.1).
5) We will then e-mail back the user's activation code
6) Restart the program (with CAPS locked) and enter the activation code in the Deep-Paratie activation window (select the option Unlock application) (Figure 1.6.2).
(please pay attention not to paste the activation code with any additional space characters)
7) Select Enter Application and press Continue.

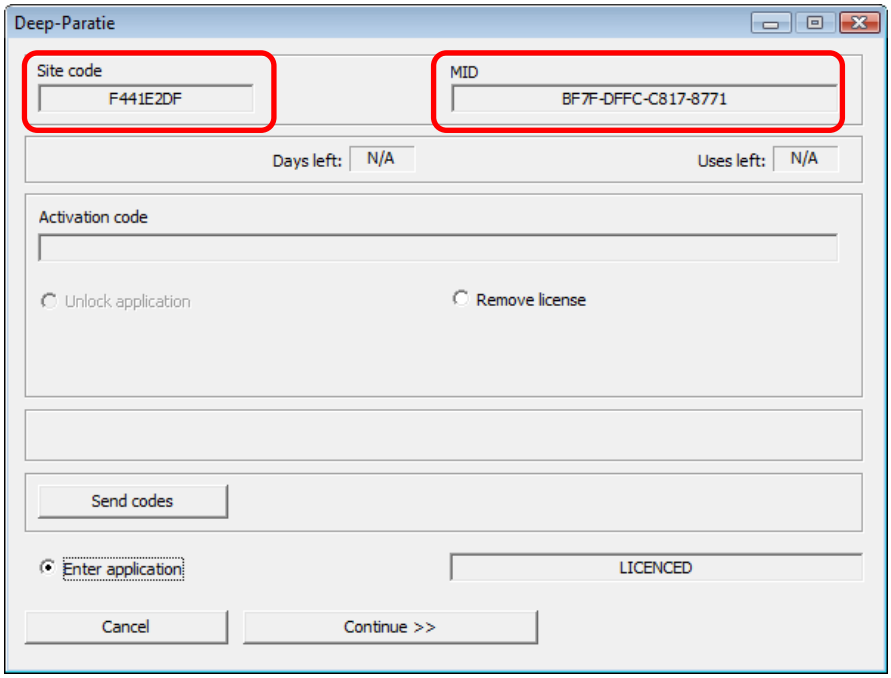


Figure 1.6.1: The DeepXcav activation window – SITE and MID codes**.**

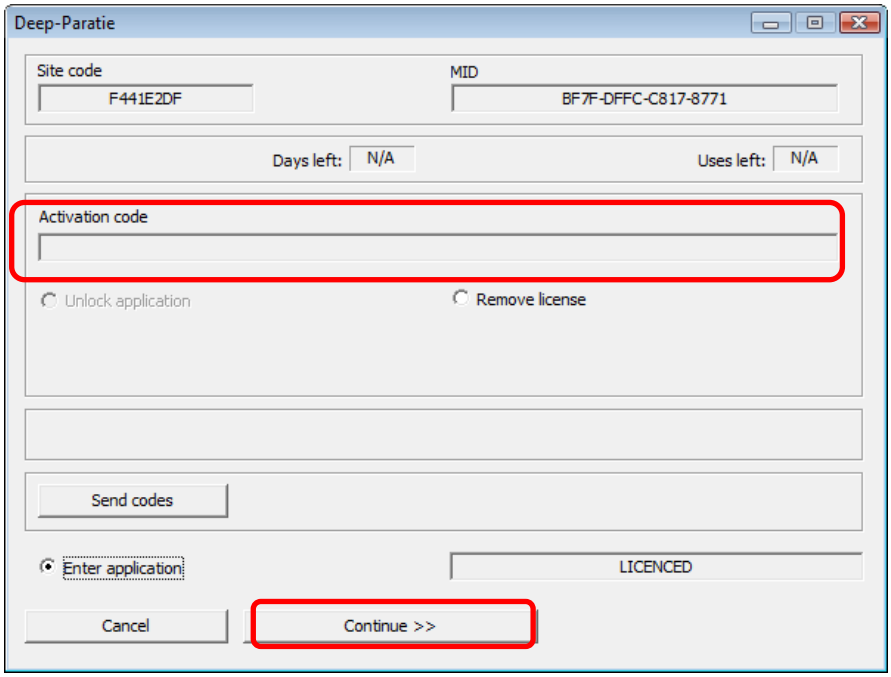


Figure 1.6.2: The activation code area**.**

This procedure is used to activate all DeepXcav's modules.

1.6.2 Activating the network license

In order to activate a network license, the following steps must be followed by the network server and by the end users:

SERVER SIDE INSTRUCTIONS

1. Install the driver for the USB key on the SERVER. The HASP driver is available from :

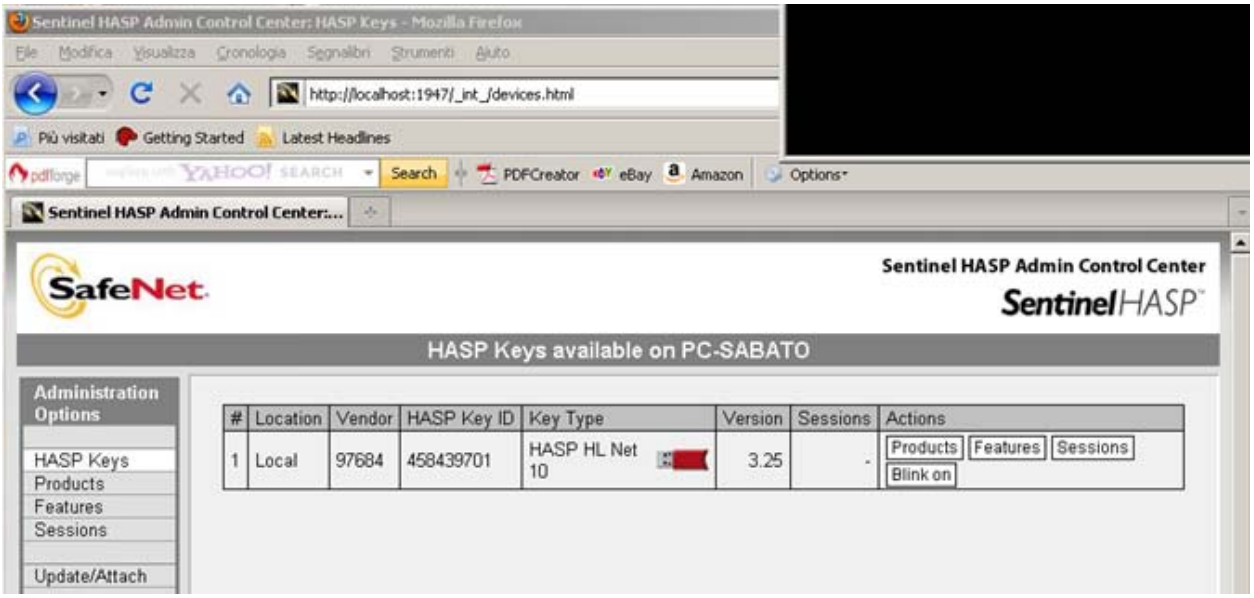
   http://www.deepexcavation.com/downloads/Sentinel_HASP_Run-time_setup.zip

   You will need to extract this file and perform the installation.

2. The HASP driver setup in included in the following link: http://localhost:1947/

   Run it on the server (the PC with the dongle).

After the driver installation, the user can open a browser and enter the url http://localhost:1947/ . This will bring up a window as it appears in the following image:


Sentinel HASP Admin Control Center: HASP Keys - Mozilla Firefox
File Modifica Visualizza Cronologia Segnalibri Strumenti Aiuto
http://localhost:1947/_int_/devices.html
Più visitati Getting Started Latest Headlines
Search PDFCreator eBay Amazon Options
Sentinel HASP Admin Control Center:...
SafeNet.
Sentinel HASP Admin Control Center
SentinelHASP
HASP Keys available on PC-SABATO
Administration Options
HASP Keys
Products
Features
Sessions
Update/Attach
Location Vendor HASP Key ID Key Type Version Sessions Actions
1 Local 97684 458439701 HASP HL Net 10 3.25 - Products Features Sessions Blink on

CLIENT SIDE INSTRUCTIONS (User PC)

3. Install the driver for the USB key on the CLIENT. The HASP driver is available from :

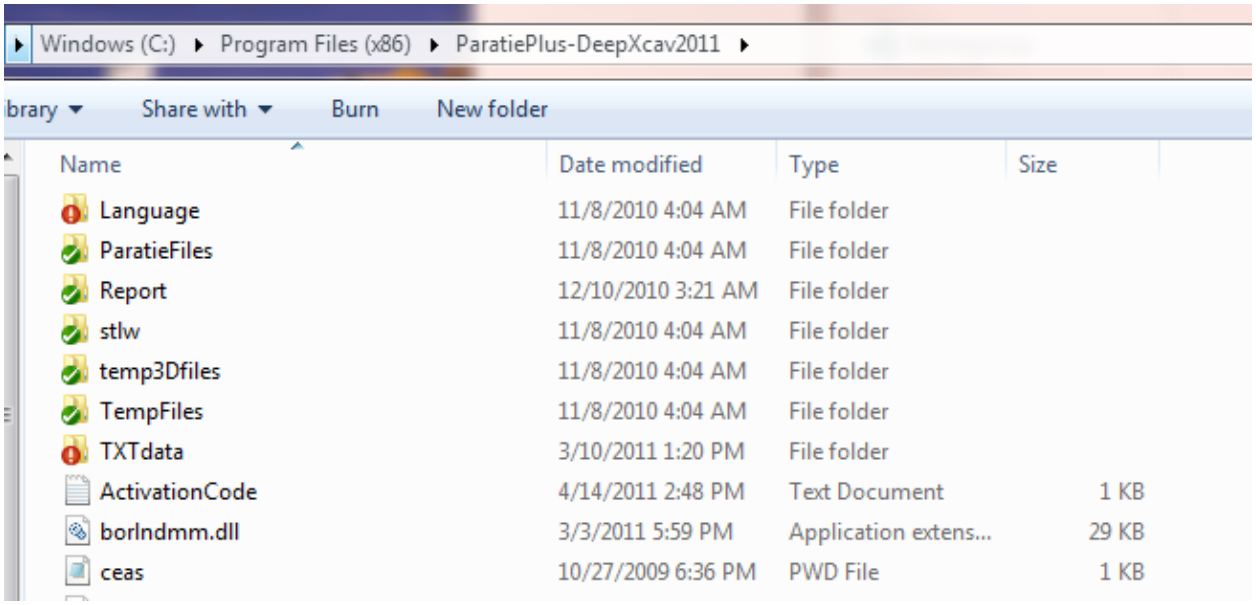
http://www.deepexcavation.com/downloads/Sentinel_HASP_Run-time_setup.zip

You will need to extract this file and perform the installation.

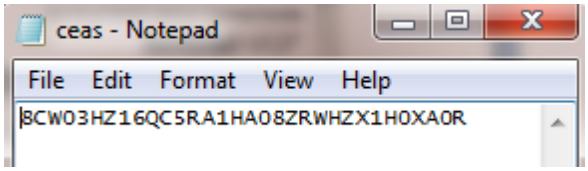
4. Install the DeepXcav program in the client pc. The program is available from:

http://www.deepexcavation.com/downloads/Setup_DeepXcav_2011.exe

5. Your distributor will provide to you a ceas.pwd file. Please place a copy of the ceas.pwd file in the client side, where DeepXcav is installed. The ceas.pwd file contains your HASP key passwords (it is recommended that you keep a copy of the file for your records).


Windows (C:) ▸ Program Files (x86) ▸ ParatiePlus-DeepXcav2011 ▸
brary ▾
Share with ▾
Burn
New folder
Name
Date modified
Type
Size
Language 11/8/2010 4:04 AM File folder
ParatieFiles 11/8/2010 4:04 AM File folder
Report 12/10/2010 3:21 AM File folder
stlw 11/8/2010 4:04 AM File folder
temp3Dfiles 11/8/2010 4:04 AM File folder
TempFiles 11/8/2010 4:04 AM File folder
TXTdata 3/10/2011 1:20 PM File folder
ActivationCode 4/14/2011 2:48 PM Text Document 1 KB
borlndmm.dll 3/3/2011 5:59 PM Application extens... 29 KB
ceas 10/27/2009 6:36 PM PWD File 1 KB

Sample ceas.pwd file containing one password file.



Please ensure that the firewall on the server is disabled so that the client PC can see the server ports. Otherwise, the user must open the 'port' used by the driver and network key.

CHAPTER 2: USING DeepXcav

2.1 Deep Excavations Background

2.1.1. Background

A deep excavation is typically defined as an excavation in soil or rock that is deeper than 15 ft (4.5 m). Deep excavations require careful design and planning especially when constructed in congested urban areas. Selecting and designing an appropriate earth retaining and support system can have significant impact on cost, time, and performance.

Figure 2.1.1: Diaphragm wall excavation supported by soil anchors**.**

Deep excavations involve two main systems a) The Retaining System that contains earth and water from entering directly into the excavated site, and b) The Support System (or bracing system) that contains the resist forces generated by soil, surcharges, and water. Cantilever excavations do not require bracing.

The term "Retaining System" for a deep excavation refers to the structural system that retains soil and water and prevents it from collapsing into the open cut. Many types of retaining systems exist. The selection of the proper retaining system depends on a wide range of factors such as: Economical, soil conditions, protection of adjacent structures, ease of construction, environmental issues and so on. Typical retaining systems include soldier pile and lagging, sheet piling, secant pile or tangent pile walls, soil mix walls, and diaphragm walls (also known as slurry walls in the U.S.).

The role of support systems is to provide lateral bracing for retaining walls. Support systems can be subdivided into external and internal depending on the load transfer mechanism. External supports (namely tiebacks) work by transferring lateral excavation loads beyond the active zone of soil movements (retained side). On the other hand, internal supports such as struts, rakers, or floor slabs, transfer lateral loads within the excavation (across opposing walls or to other internal structures). In all cases, support and retaining systems have to work closely together in order to guarantee a high level of performance.

More detailed descriptions of typical retaining and support systems are available in following sections.

2.1.2. RETAINING SYSTEMS FOR DEEP EXCAVATIONS

2.1.2. A. Soldier Pile and Lagging Walls

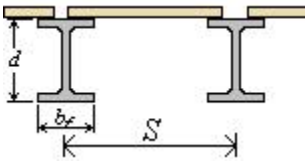


Figure 2.1.2: Soldier Pile & Timber Lagging Walls.

Soldier pile and lagging walls are some of the oldest forms of retaining systems used in deep excavations. These walls have successfully being used since the late 18th century in metropolitan cities like New York, Berlin, and London. The method is also commonly known as the "Berlin Wall" when steel piles and timber lagging is used. Alternatively, caissons, circular pipes, or concrete piles can also be used as soldier piles (but at an increased cost). Timber lagging is typically used although reinforced concrete panels can also be utilized for permanent conditions. Soldier pile and lagging walls are formed by:

1. Constructing soldier piles at regular intervals (6 ft to 12 ft, typically)

2. Excavating in small stages and installing lagging.

3. Backfilling and compacting the void space behind the lagging.

Moment resistance in soldier pile and lagging walls is provided solely by the soldier piles. Passive soil resistance is obtained by embedding the soldier piles beneath the excavation grade. The lagging bridges and retains soil across piles and transfers the lateral load to the soldier pile system.

Soldier pile and lagging walls are the most inexpensive systems compared to other retaining walls. They are also very easy and fast to construct. The major disadvantages of soldier pile and lagging systems are:

1. They are primarily limited to temporary construction.

2. Cannot be used in high water table conditions without extensive dewatering.

3. Poor backfilling and associated ground losses can result in significant surface settlements.

4. They are not as stiff as other retaining systems.

5. Because only the flange of a soldier pile is embedded beneath subgrade, it is very difficult to control basal soil movements.

2.1.2. B. Sheet Pile Walls

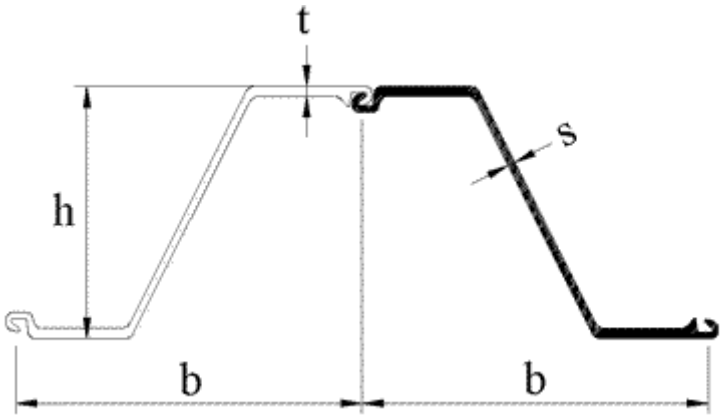


Figure 2.1.3: Sheet Pile Section.

Sheet pile walls are constructed by driving prefabricated sections into the ground. Soil conditions may allow for the sections to be vibrated into ground instead of it being hammer driven. The full wall is formed by connecting the joints of adjacent sheet pile sections in sequential installation. Sheet pile walls provide structural resistance by utilizing the full section. Steel sheet piles are most commonly used in deep excavations, although reinforced concrete sheet piles have also been used successfully.

Steel sheet piling is the most common because of its several advantages compared to other materials:

1. Provides high resistance to driving stresses.

2. Light weight

3. Can be reused on several projects.

4. Long service life above or below water with modest protection.

5. Easy to adapt the pile length by either welding or bolting

6. Joints are less apt to deform during driving.

2.1.2. C. Secant Pile Walls

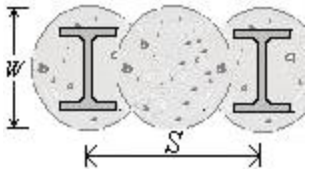


Figure 2.1.4: Typical Secant Pile Wall Section.

Secant pile walls are formed by constructing intersecting reinforced concrete piles. The piles are reinforced with either steel rebar or with steel beams and are constructed by drilling under mud. Primary piles are installed first with secondary piles constructed in between primary piles once the latter gain sufficient strength. Pile overlap is typically in the order of 3 inches (8 cm). In a tangent pile wall, there is no pile overlap as the piles are constructed flush to each other. The main advantages of secant or tangent pile walls are:

1. Increased construction alignment flexibility.
2. Increased wall stiffness compared to sheet piles.
3. Can be installed in difficult ground (cobbles/boulders).
4. Less noisy construction.

The main disadvantages of secant pile walls are:

1. Verticality tolerances may be hard to achieve for deep piles.
2. Total waterproofing is very difficult to obtain in joints.
3. Increased cost compared to sheet pile walls.

2.1.2. D. Soil Mix Walls

Various methods of soil mixing, such as mechanical, hydraulic, with and without air, and combinations of both types have been used widely in Japan for about 20 years. Soil mixing has been used for many temporary and permanent deep excavation projects including the Central Artery project in Boston. Known methods include as Jet Grouting, Soil Mixing, Cement Deep Mixing (CDM), Soil Mixed Wall (SMW), Geo-Jet, Deep Soil Mixing, (DSM), Hydra-Mech, Dry Jet Mixing (DJM), and Lime Columns. Each of these methods aims at finding the most efficient and economical method to mix cement (or in some cases fly ash or lime) with soil and transform soil to become more like a soft rock.

Mechanical soil mixing is performed using single or multiple shafts of augers and mixing paddles. The auger is slowly rotated into the ground, typically at 10-20 rpm, and advanced at 2 to 5 ft (0.5 to 1.5 m) per minute.

Cement slurry is pumped through the hollow stem of the shaft(s) feeding out at the tip of the auger as the auger advances. Mixing paddles are arrayed along the shaft above the auger to provide mixing and blending of the slurry and soil. Slurry lubricates the tool and assists in the breaking up of the soil into smaller pieces. Spoils come to the surface since fluid volume is being introduced into the ground. These spoils comprise cement slurry and soil particles with similar cement content as what remains in the ground. After final depth is reached, the tools remain on the bottom of the hole, rotating for about 0.5 to 2 minutes for complete mixing. At this point, the tools are raised while continuing to pump slurry at a reduced rate. Withdrawal is typically at twice the speed of penetration, 4 ft to 10 ft (1 m to 3m) per minute.

Steel beams are typically inserted in the fresh mix to provide reinforcement for structural reasons. A continuous soil mix wall is constructed by overlapping adjacent soil mix elements. Soil mix sections are constructed in an alternating sequence with primary elements being formed first and secondary elements following once the first have gained sufficient strength.

The soil mix method can be very effective at providing very stiff and waterproof retaining systems. However, it is rather limited to medium and large-scale projects because of its high mobilization costs. Insufficient mix strength may result when mixing organic soils unless a high replacement ratio is maintained. Other issues such as difficulties in maintaining consistent compressive strengths throughout the section of a soil mix wall can also emerge.

2.1.2. E. Diaphragm Walls (Slurry walls US)

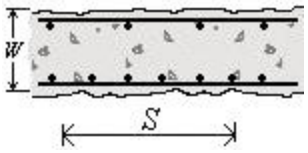


Figure 2.1.5: Typical Diaphragm Wall Section.

The continuous diaphragm wall (also referred to as slurry wall) is a structure formed and cast in a slurry trench (Xanthakos, 1994). The trench is initially supported by either Bentonite polymer based slurries. The term "diaphragm walls" refers to the final condition when the slurry is replaced by tremied concrete that acts as a structural system either for temporary excavation support or as part of the permanent structure. This construction sequence is illustrated in Figure 1.The term slurry wall is also applied to walls that are used as flow barriers (mainly in waste containment), by providing a low permeability barrier to contaminant transport.

Slurry wall technology hinges on specialized equipment for excavating slurry trenches. The simplest type of trenching equipment is the mechanical clamshell attached on a kelly bar. Individual contractors have developed their own specialized trenching equipment like hydraulic clamshells, fraise or hydromills (sample manufacturers: Icos, Bauer, Casagrande, Case Foundation, Rodio etc).

The first diaphragm walls were tested in 1948 and the first full scale slurry wall was built by Icos in Italy in 1950 (Puller, 1996) with Bentonite slurry support as a cut-off wall. Icos constructed the first structural slurry wall in the late 1950s for the Milan Metro (Puller, 1996). Slurry walls were introduced in the US in the mid 1960s by European contractors. The first application in the US was in New York City [1962] for a 7m diameter by 24m deep shaft (Tamaro, 1990), that was followed by the Bank of California in San Francisco (Clough and Buchignani, 1980), the CNA building in Chicago (Cunningham and Fernandez, 1972), and the World Trade Center in New York (Kapp, 1969, Saxena, 1974). The majority of diaphragm wall projects in the US are located in six cities Boston, Chicago, Washington DC, San Francisco and New York.

Diaphragm walls are extensively used in the Central Artery/Tunnel project (CA/T) in Boston, Massachusetts. Work in the CA/T involves many cut and cover tunnels constructed under the existing artery. Some of the deepest T-slurry walls, extending 120' below the surface have been constructed for the Central Artery (Lambrechts et al., 1998).

2.1.3 SUPPORT SYSTEMS FOR DEEP EXCAVATION:

2.1.3. A. Tiebacks/Rock Anchors

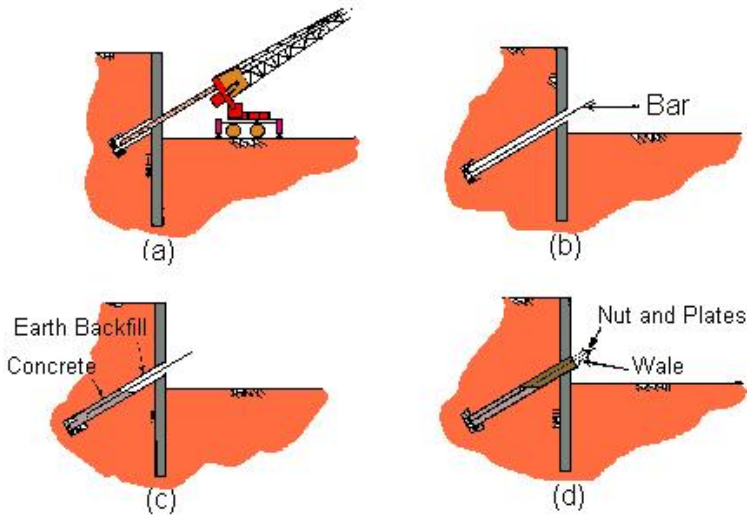


Figure 2.1.6: General installation sequence of tiebacks.

Anchored walls have become popular in braced excavations because of a) the substantial progress in the technology and availability of high-capacity anchor systems, and b) the absence of interior obstructions that permit uninterrupted earth moving and thus improve the construction conditions of the underground portion of a building (Xanthakos, 1994). In some projects tiebacks have been used in combination with rakers and soil berms and/or corner braces (Gnaedinger et al., 1975). Tieback anchors comprise a barrel anchorage located either in a bearing layer which is tensioned at the front face of the wall. The part of the anchor that transfers the force to the surrounding soil is frequently called the "fixed length", while the "free length" transmits forces from the fixed length through the anchor head to the slurry wall.

In order to minimize wall movement and ground settlement, tieback anchors are designed to achieve the highest stiffness possible within economical considerations. In urban cities like Boston, Chicago, New York, and Washington where land is precious such deep excavations are more common. Tieback capacity depends on the vertical and horizontal spacing of anchors and on surcharge conditions. Prestress levels typically range from 40 to 250 kips when the grouted portion of tiebacks is within soil, higher loads are used when the ties are located in bedrock.

Typical tieback spacing ranges from 7ft to 13ft in the vertical, and from 5ft to 15ft in the horizontal direction. Tieback capacity is reduced if the spacing is too close due to interference between adjacent grouted zones.

Often the tiebacks are used only for temporary excavation support, while the basement floors provide permanent lateral earth support. In such projects the tiebacks are detensioned when the basement floors have gained sufficient strength. The basement floors should be designed to resist permanent lateral earth pressures, since stress transfer from the tiebacks to the floor system will take place when the ties are detensioned. This stress transfer has reportedly caused long-term cracking of many the basement floors.

Tieback installation follows a predetermined sequence as to minimize soil movements and speed the excavation construction. The excavation is carried a couple of feet below the tieback to enable access for the drill rig. Further excavation occurs only after prestressing and proof-testing of the anchors. The process can be repeated for additional levels of tiebacks. Building codes require that all tiebacks are proof-tested to an excess percentage of their final lock-off load, which usually ranges from 120 to 150% of the final lock-off load. Regroutable tiebacks are most commonly used because their capacity can be increased by regrouting (to meet test requirements) without having to drill a new anchor hole.

A tieback is made by first drilling a hole with an auger and then placing a bar (tendon) in the hole, concrete is then poured in the hole and the connection with wall is made. Different types of augers are used to drill the tieback holes. The choice of the drilling method depends on the soil/rock conditions on the site.

Drilling should be done carefully since inadequate procedures can cause significant soil losses. The byproduct of drilling is removed by flushing the hole with air, water, or slurry. Air is most efficient in dry ground, but it requires special attention because it can become entrapped during drilling, building up zones of high pressure in the soil that can eject material for several feet and at high speeds (potentially injuring workers). Water flushing is best used in sticky clayey soil, and it also cleans the sides of the hole by its sweeping action, providing a stronger bond at the grout-anchor interface. Bentonite slurry flushing works the best since it keeps particles in suspension, while the sealing action keeps the hole from collapsing.

Significant soil losses through the tiebacks cause significant settlements even if the retaining walls do not move towards the excavation. In granular soils the drilled hole must be cased to avoid collapse.

Some tieback creep can be expected especially if the ties are very short and the fixed length of the tie is within soft ground. For stability reasons, the fixed anchor should be located beyond

the active zone of movements. As a result, tieback anchors may not be an option at sites congested where there are adjacent underground utilities or when adjacent owners do not grant permission to drill them under their properties.

Special attention should be given to the waterproofing details at the anchor heads and at the tieback holes. Significant leakage can be caused by inadequate water stopping details at these locations.

2.1.3. B. Cross-lot/Internal Bracing

Figure 2.1.7: A cross-lot project.

Cross-lot or internal bracing transfers the lateral earth (and water pressures) between opposing walls through compressive struts. Rakers resting on a foundation mat or rock offer another internal bracing alternative. Typically the struts are either pipe or I- beam sections and are usually preloaded to provide a very stiff system. Installation of the bracing struts is done by excavating soil locally around the strut and only continuing the excavation once preloading is complete. The struts typically rest on a series of wale beams that distribute the strut load to the diaphragm wall.

Pre-loading ensures a rigid contact between interacting members and is accomplished by inserting a hydraulic jack as each side of an individual pipe strut between the wale beam and a special jacking plate welded to the strut. The strut load can either be measured with strain

gages or can be estimated using equations of elasticity by measuring the increased separation between the wale and the strut.

In some earlier projects the struts were not preloaded, and as a result when the excavation progressed deeper the soil and the wall movements were large (C1). Thus it has become standard practice to preload struts in order to minimize wall movements. Cross-lot bracing makes sense in narrow excavations (60ft to 120ft) when tieback installation is not feasible. The struts can bend excessively under their own weight if the excavation spacing is too large. In addition, special provisions have to taken to account for thermal expansion and contraction of the struts.

The typical strut spacing is in the range of 15ft, both in the vertical and the horizontal direction. This is larger than the typical spacing when tiebacks are used, because the pre-loading levels are much higher. A clear benefit of using struts is that there are no tieback openings in the slurry wall, thus eliminating one source of leakage.

2.1.3. C. Top/Down Construction

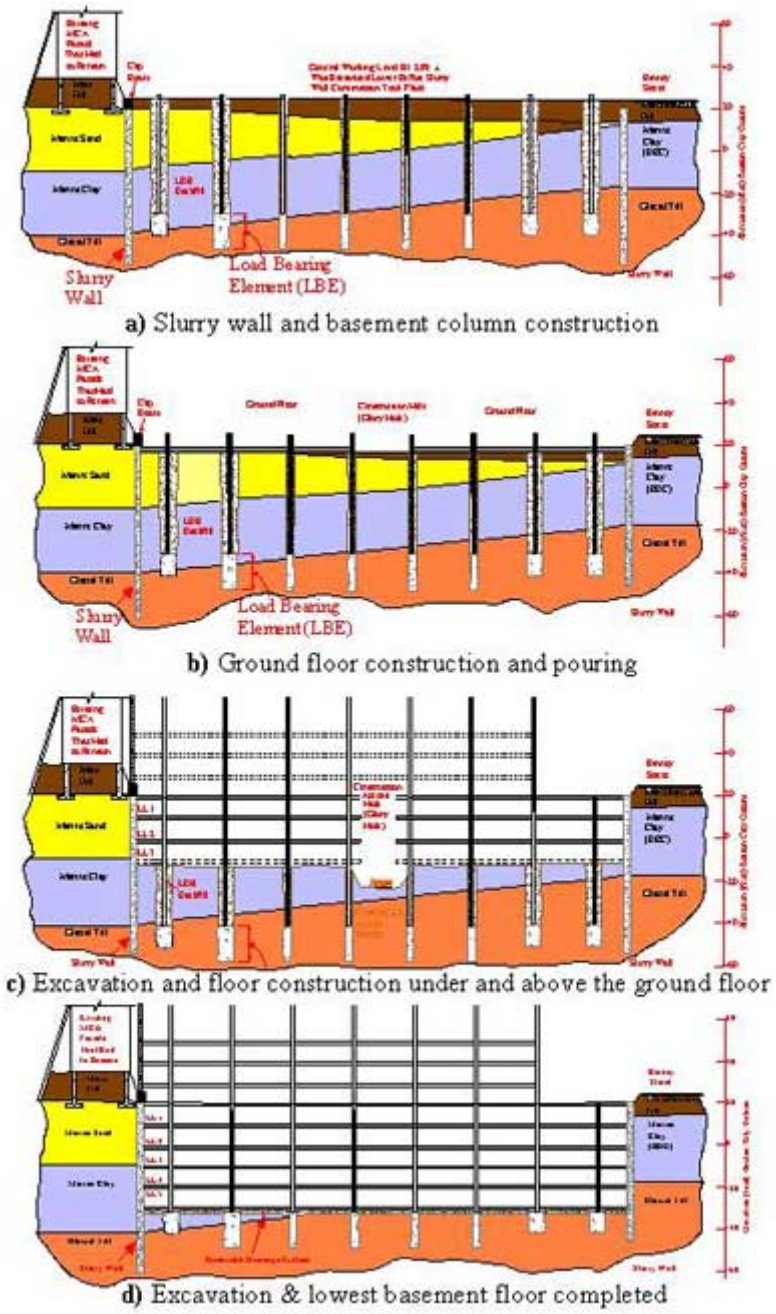

Slurry Wall
Load Bearing Element (LBE)
a) Slurry wall and basement column construction
Slurry Wall
Load Bearing Element (LBE)
b) Ground floor construction and pouring
c) Excavation and floor construction under and above the ground floor
d) Excavation & lowest basement floor completed

Figure 2.1.8: Typical Top/Down Sequence.

Top/down or up/down construction methods are another method for constructing deep excavations. In this case the basement floors are constructed as the excavation progresses. The top/down method has been used for deep excavation projects where tieback installation was not feasible and soil movements had to be minimized. The general top/down construction sequence is shown in the figure above. The Post Office Square Garage in Boston (7-levels deep) is one of the best-instrumented and documented top/down projects in the US (Whittle, et al., Whitman et al., 1991).

The sequence construction begins with retaining wall installation and then load-bearing elements that will carry the future super-structure. The basement columns (typically steel beams) are constructed before any excavation takes place and rest on the load bearing elements. These load bearing elements are typically concrete barrettes constructed under slurry (or caissons). Then the top floor slab is constructed with at least on construction (glory) hole left open to allow removal of spoil material.

The excavation starting at the glory hole begins once the top floor has gained sufficient strength. Soil under the top basement floor is excavated around the basement columns to slightly lower than the first basement floor elevation in order to allow for the installation of the forms for the first level basement slab. Glory holes are left open within each newly formed basement floor slab and the procedure is repeated. Each floor rests on the basement columns that were constructed earlier.

2.2 Using DeepXcav

DeepXcav is a user-friendly software program and includes powerful features and versatile options. In DeepXcav we can work with many design sections of an excavation. In a sense, a design section is a design scenario. Each design section can be independent or can be linked to a parent model. This way, multiple conditions can be examined simultaneously. The main interface is shown in Figure 2.2. The general philosophy in creating an excavation model in DeepXcav is:

1) Specify the global coordinates.

2) Specify the soil types and properties.

3) Specify the layers.

4) Create a generalized water table.

5) Specify the retaining wall system (soldier pile, sheet pile, secant pile, tangent pile, and diaphragm wall options).

6) Create a database of support members (tiebacks, struts, or concrete slabs).

7) Add a stage and draw in new supports.

8) Modify the stage elevations.

9) Specify load cases or design approach combinations.

10) Analyze the project.

11) Perform slope stability analysis.

The general tabs that appear on the top of the program have the following functions.

1. General: This tab includes general information about the project, model limits and some examples that may help you get started using the program.

2. Properties: This tab contains various information about Borings, soils, wall sections, Structural sections of supports such as anchors, slabs, struts and Helical anchors for tiebacks. In addition there is information and settings about the structural materials being used.

3. Model: Here we can define borings, number of walls used, surface and water elevations, add or delete stages and apply some other advanced features.

4. Loads and Supports: Here we can modify/edit options for supports and external loads..

5. Seismic: In this tab we can include and edit seismic effects.

6. Analysis: In this tab we can define the type of the analysis performed (limit equilibrium, elastoplastic or both).

7. Slope: This is an extra module which includes options for slope stability.

8. Stability +: In this tab we can control options for the calculation of wall embedment safety factors, surface settlements and Clough basal stability method.

9. Design: In this tab we can control the structural design codes and other options for calculating the design capacities of walls and supports.

10. Results: In this tab we can select to present results directly to the screen after the analysis is performed.

11. Report: In this tab we can select options for generating output reports, or viewing calculation progress files..

12. View: In this tab we can modify various view options or generate a top view of the model.

13. Optimize: This tab provides options for optimizing walls and supports after an analysis is completed.

14. Help: This tab provides links to help and technical manuals.

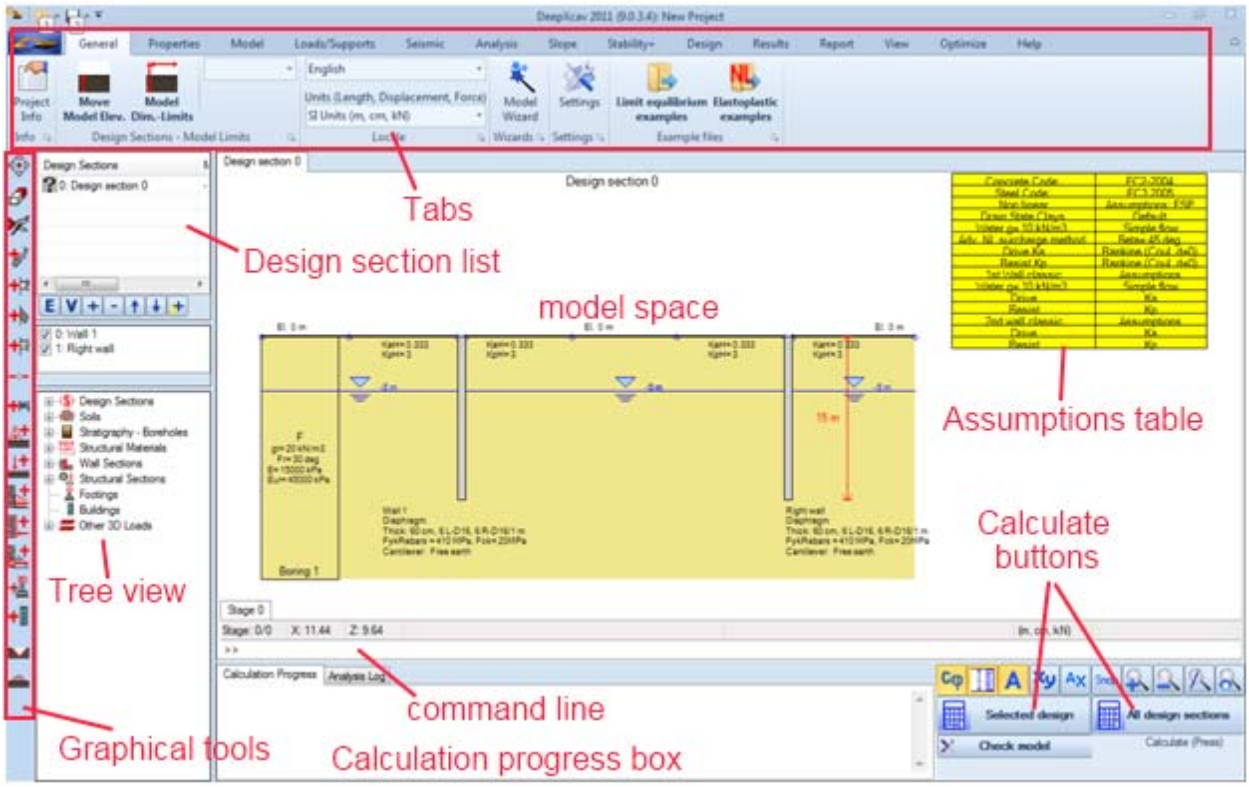


Figure 2.2: General DeepXcav 2011 Interface

2.3.1 DeepXcav Toolbar Functions

The following section provides a detailed list of all toolbar functions. The first tab group to encounter contains the following options:

- **Start button**

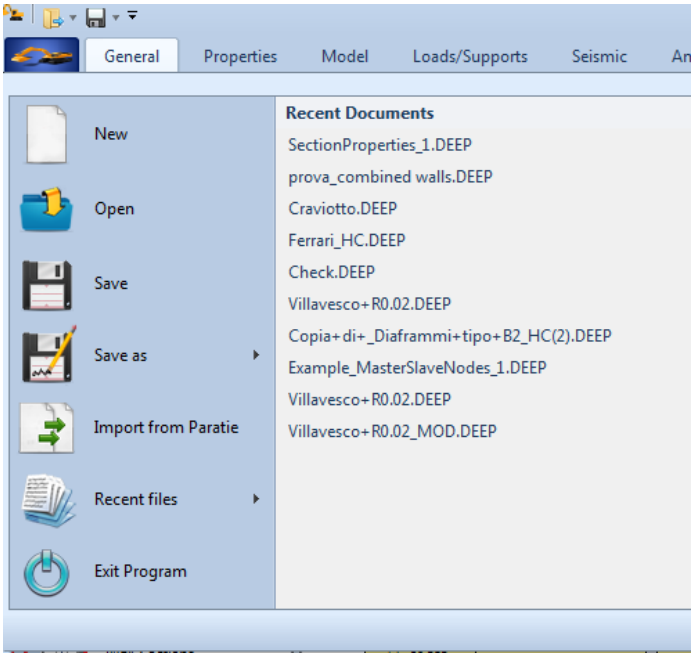


Figure 2.3.1: Main button

Tool	Description
	Create a new project.
	Open an existing project.
	Saves the project.
	Save a project with different name or destination folder, or in an older version.
	Import a file created with the Paratie program (Italian users only).
	List of recently created or modified projects.
	Exits the program.

A vertical toolbox is available on the left screen side. The available tools are described in the table below:

Tool	Description
	Select tool – drag and move objects
	Erase an item. Select an item on the model to erase.
	Splits a surface line and inserts a point.
	Adds a tieback – anchor (first click on the wall and then in the ground).
	Adds a strut (first click on the wall and then in the ground, or on the second wall).
	Adds a raker click on the wall and then in the ground).
	Adds a slab (first click on the wall and then define the end point of the slab).
	Adds a fixed support (click on the wall to add a fixed support).
	Adds a spring (click on the wall to add a spring).
	Adds a surface surcharge (define the start and end point of the surcharge).
	Adds a surface line load (click a surface point to add a point load).
	Adds a surcharge on the wall (define two wall points to add a surcharge).
	Adds a line load on the wall (define a wall point to add a wall point load)
	Adds a prescribed condition at a wall (click on the wall to add a prescribed condition).
	Adds a footing load (3D) (define a point where to install a footing load).
	Creates a new building (define a point where to install a building).
	Performs an excavation (click on screen to define the excavation limits).
	Performs a backfill operation (click on screen to define the backfill limits).

On the top left side of the program, right under the design section list appears a toolbar that helps the user deal with the design sections (Figure 2.3.2). The icons are presented and described in the table below:

Tool	Description
	Edit the name of the selected design section
	Generate a new view of the current design section
	Add a new design section
	Delete design section
	Move design section up on the list
	Move design section down on the list
	Add a new design section (empty – including only stage 0)

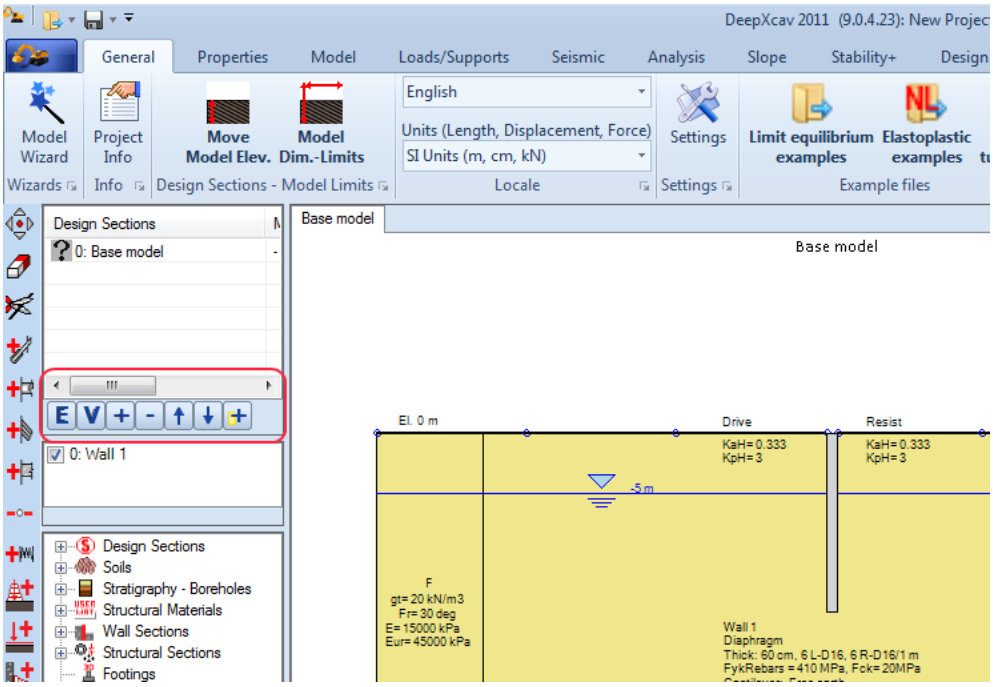


Figure 2.3.2: Design section toolbars

On the bottom right corner of the program there is a toolbar that helps the user perform calculations and control viewing options.

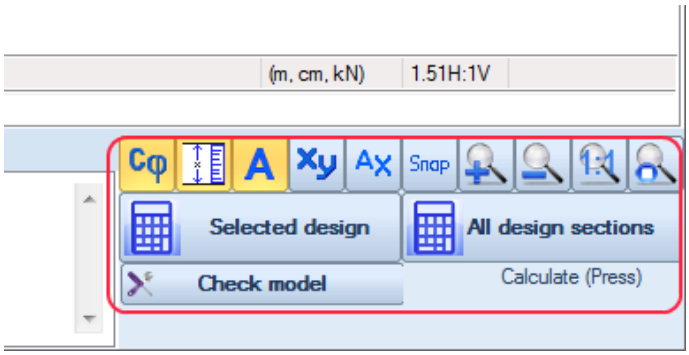
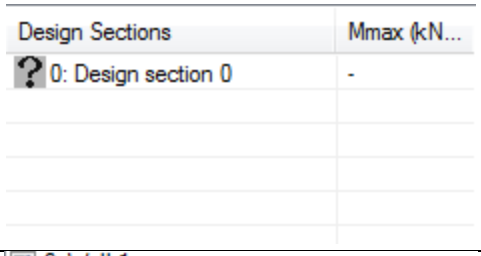
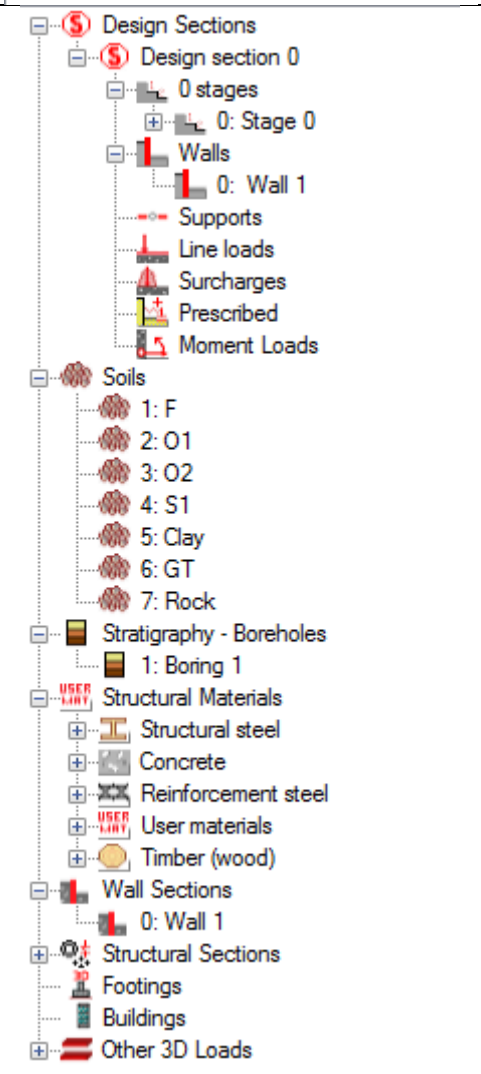


Figure 2.3.3: View and calculate tools

Tool	Description
Cφ	Show soil properties
	Show dimensions
A	Show assumptions
Xy	Show point coordinates
Ax	Show axis
Snap	Turn the snap on or off
	Zoom in
	Zoom out
1:1	Zoom to 1 to 1 horizontal to vertical scale
	Zoom Original (all)
Selected design	Calculate the selected design section
All design sections	Calculate all design sections
Check model	Perform an integrity check of the model

2.3.2 Wall List, Design Section List, and Project Tree View

DeepXcav offers features that include multiple design sections and a tree-style project view. The tree view enables the user to quickly access vital project data, as well as visualize crucial project settings. The next table briefly describes the functionality of the Wall list, Design Section List, and Tree View items.

	Selects current design section, shows available design sections.
	Selects on which wall on current design section the settings should be applied.
	Shows available design sections Shows used wall types Lists supports (right click to add or erase) Line loads (right click to add or erase) Surcharges (right click to add or erase) Prescribed conditions (right click to add or erase) Moment loads (right click to add or erase) Available soil types (by clicking the user can select which soil's properties to modify) Available boreholes (by clicking the user can select which borehole's properties to modify) Structural materials (by clicking the user can select to change the properties of the structural materials) Available wall sections (By clicking the user can see the available wall sections, delete them, activate and deactivate them) Available structural sections (buildings and footings) Other 3D loads (Area loads, 3D point loads and vehicle loads)

2.4 General menu

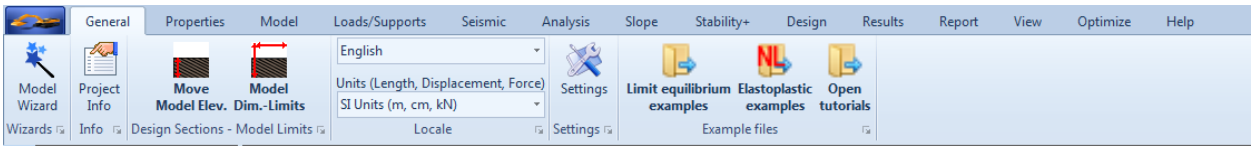
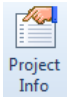


Figure 2.4.1: Project info, Model limits, Locale, Wizards, Settings, Example files

- **Project Info:** by pressing the button 

, we can change the project, file, company and engineer name

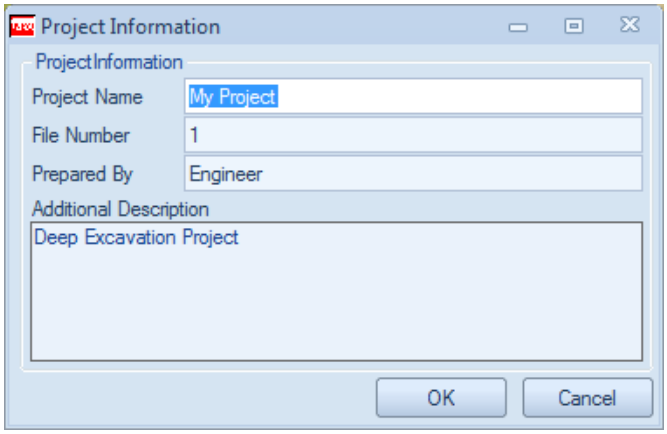


Figure 2.4.2: Project information dialog.

- **Move model elevation:** by pressing the button

, we can change the model elevation by entering a new top of wall elevation.

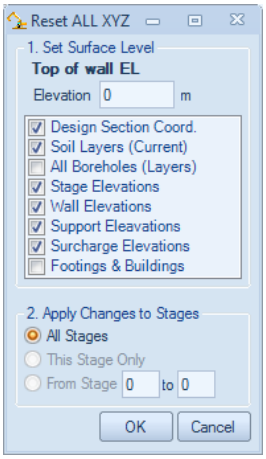


Figure 2.4.3: Model Elevation dialog.

The user can choose the objects to be affected by the change in elevation. These are:

The design section coordinates
The soil layers elevation (of current borehole)
All the soil layer elevations of all boreholes
Elevations of all stages
Elevations of walls
Elevations of all supports
Elevations of all surcharges
Elevations of all footings and buildings

The top of the wall is used as point of reference for the change of elevations.

- **Model Dimension - Limits:** by pressing the button

, we can change the Design Section name, the used Stratigraphy or Boring and the Model Limits to create a nice view of the model.

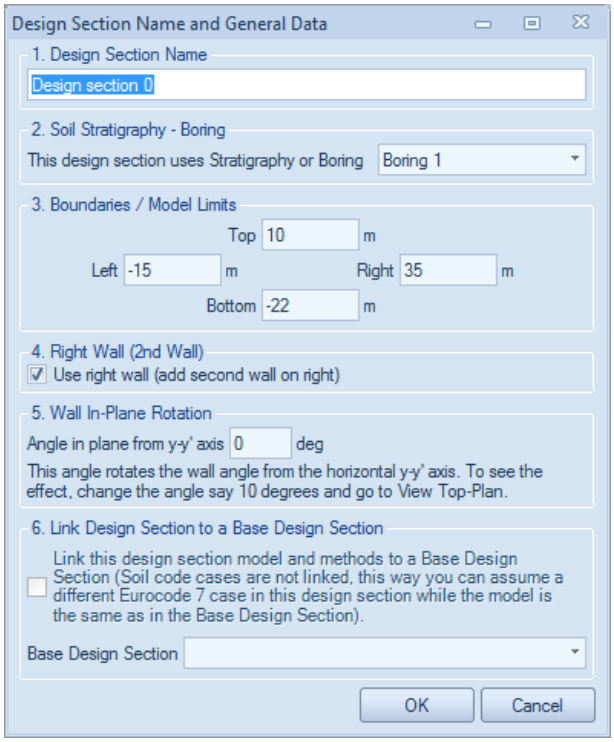
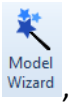

Design Section Name and General Data
1. Design Section Name
Design section 0
2. Soil Stratigraphy - Boring
This design section uses Stratigraphy or Boring
Boring 1
3. Boundaries / Model Limits
Top 10 m
Left -15 m
Right 35 m
Bottom -22 m
4. Right Wall (2nd Wall)
Use right wall (add second wall on right)
5. Wall In-Plane Rotation
Angle in plane from y-y' axis 0 deg
This angle rotates the wall angle from the horizontal y-y' axis. To see the effect, change the angle say 10 degrees and go to View Top-Plan.
6. Link Design Section to a Base Design Section
Link this design section model and methods to a Base Design Section (Soil code cases are not linked, this way you can assume a different Eurocode 7 case in this design section while the model is the same as in the Base Design Section).
Base Design Section
OK
Cancel

Figure 2.4.4: Model Dimension – Limits dialog.

This dialog includes the following options:

- The design section name.
- The boring that is used in the current model.
- The model limits. Here we can define the top, bottom, left and right limits of the model. These are absolute coordinates.
- Add a second wall to the model.
- Define the angle of the wall in-plane from y'-y axis (plan view i.e. from top).
- Link selected design section to a base design section. This connects two design sections and they are developed as one, with the capability of use of different load cases in each design section.

- **Locale:** Change the language and the units used in this model.

- **Wizard:** by pressing the button 

, the Wizard dialog appears. The wizard can create a model really fast.

➢ Welcome Tab: Here the user can define the Analysis method.

Figure 2.4.5: Wizard – Welcome Tab.

The user can choose to perform a beam on elastoplastic foundations (non linear) analysis, a conventional (limit equilibrium) analysis, or both analyses.

➢ Dimensions Tab: Here we can define units and basic model dimensions.

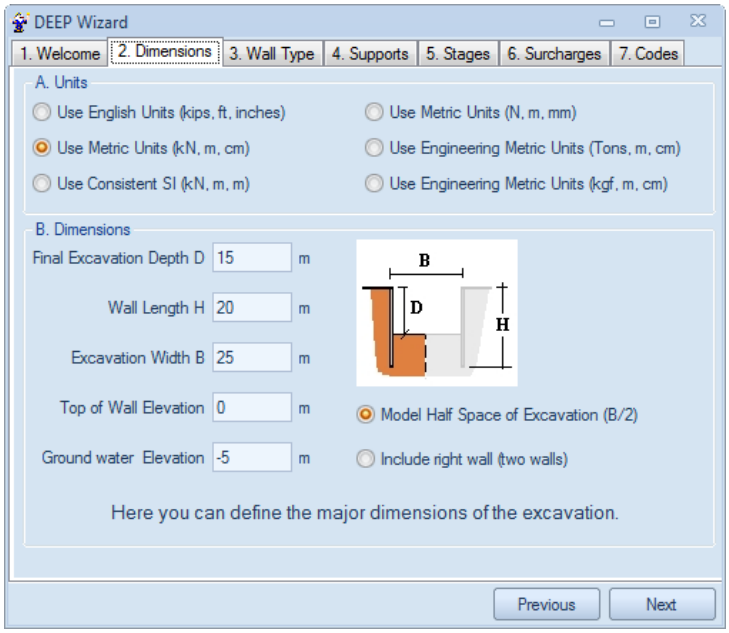


Figure 2.4.6: Wizard – Dimensions Tab.

After selecting the units, the user must define the following data:

- The final excavation depth (D).
- The wall length (H).
- The excavation width (B).
- The top of the wall elevation.
- Ground water elevation.

In addition, with the options "Model Half Space of Excavation" and Include right wall", we can choose whether to use two walls in the model or not.

➢ Wall Type Tab: Here we can define the basic wall type to be used.

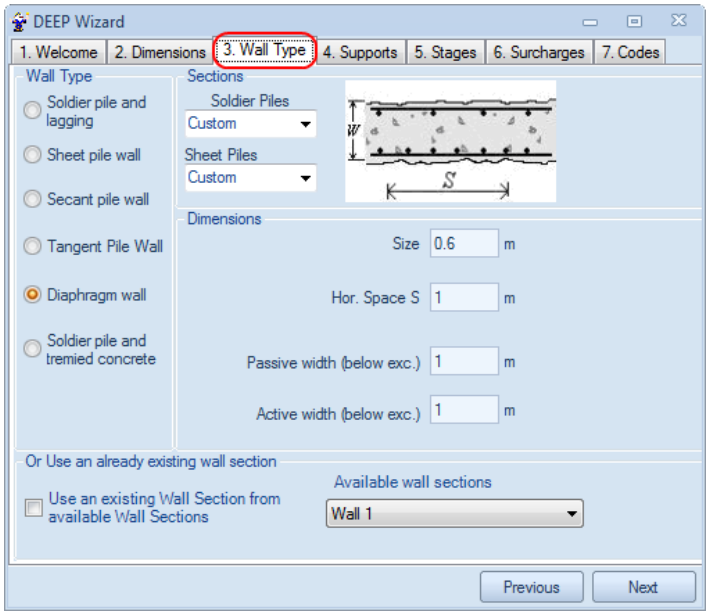


Figure 2.4.7: Wizard – Wall Type Tab.

For the definition of wall type, the user can define the following data:

- The sections of sheet or soldier piles used (if necessary).
- The size of the wall (width).
- The passive width (below excavation).
- The active width (below excavation).
- The horizontal spacing of the wall.

In addition, the user has the possibility to choose a wall from the already available list of walls created by the user.

➢ Supports Tab: Here the user can define the Support type and sections to be used.

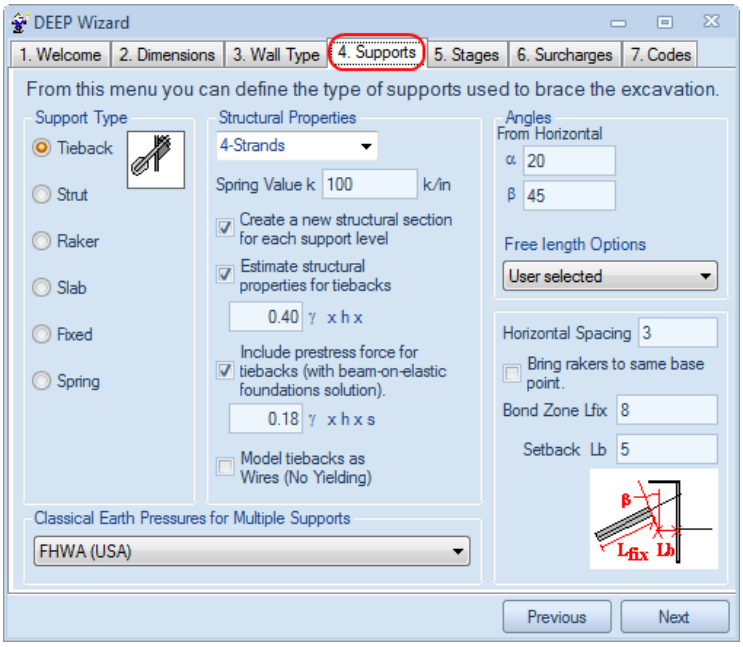


Figure 2.4.8: Wizard – Supports Tab.

With the definition of the supports, the user can define the following data:

- The structural properties of the support (section).
- Whether to create a new structural section for each support level or not.
- Whether to estimate structural section for tiebacks or not.
- Whether to include prestress for tiebacks or not (no yielding – recommended).
- Whether model tiebacks as wires or not.
- Define the angles of the supports from horizontal (when supports can be installed inclined).
- Define the free length (option available only for tiebacks).
- Define the horizontal spacing.
- Define the bond zone (fixed length, option available only for tiebacks).
- Define the setback (distance of the top of the support from the wall, available only for tiebacks and rakers).

- Stages Tab: Here the user can define the surface elevations for each construction stage. There are three main choices:

  - Automatic elevations:

With this option the program will equally space supports and automatically determine excavation depths for each stage. There are also options to include an initial stage with no excavation (recommended).

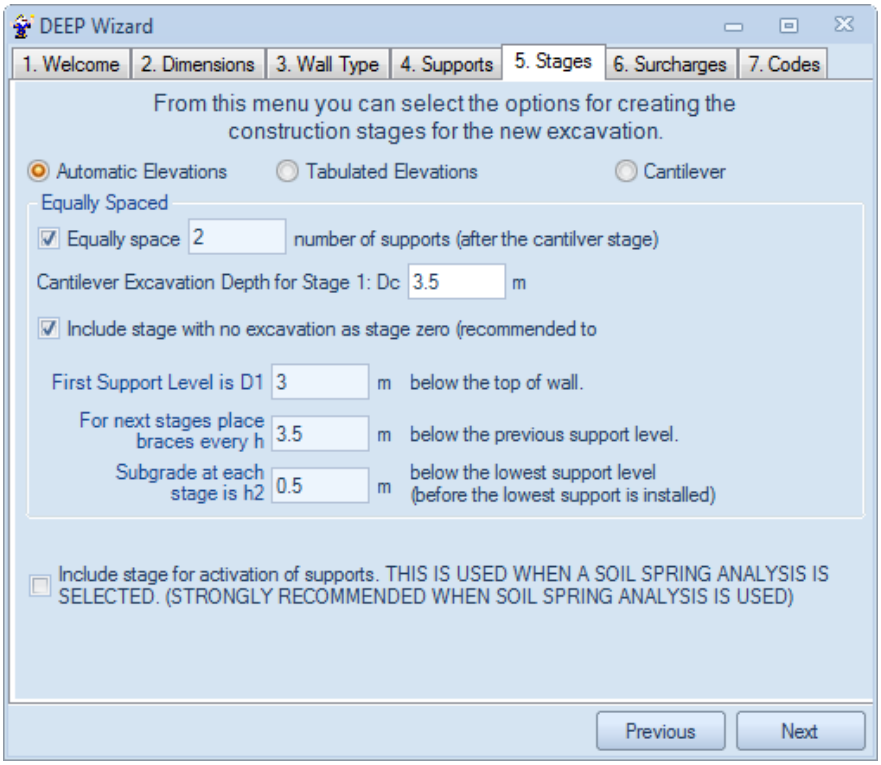

DEEP Wizard
1. Welcome
2. Dimensions
3. Wall Type
4. Supports
5. Stages
6. Surcharges
7. Codes
From this menu you can select the options for creating the construction stages for the new excavation.
Automatic Elevations
Tabulated Elevations
Cantilever
Equally Spaced
Equally space 2 number of supports (after the cantilver stage)
Cantilever Excavation Depth for Stage 1: Dc 3.5 m
Include stage with no excavation as stage zero (recommended to
First Support Level is D1 3 m below the top of wall.
For next stages place braces every h 3.5 m below the previous support level.
Subgrade at each stage is h2 0.5 m below the lowest support level (before the lowest support is installed)
Include stage for activation of supports. THIS IS USED WHEN A SOIL SPRING ANALYSIS IS SELECTED. (STRONGLY RECOMMENDED WHEN SOIL SPRING ANALYSIS IS USED)
Previous
Next

Figure 2.4.9.a: Wizard – Stages Tab, Automatic elevations.

  - Tabulated elevations:

With this option support elevations can be manually defined. The height difference between the lowest support and the subgrade can also be defined.

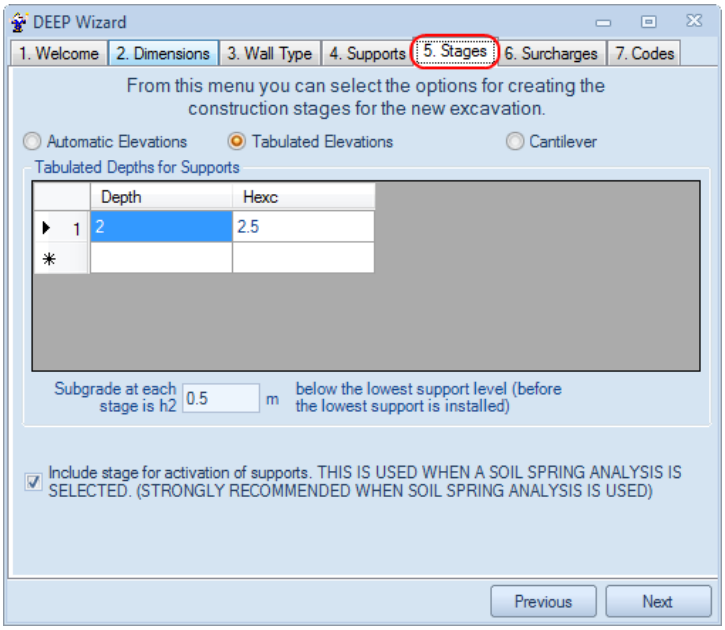


Figure 2.4.9.b: Wizard – Stages Tab, Tabulated elevations.

- Cantilever:

Select this option to create a cantilever excavation (with no supports).

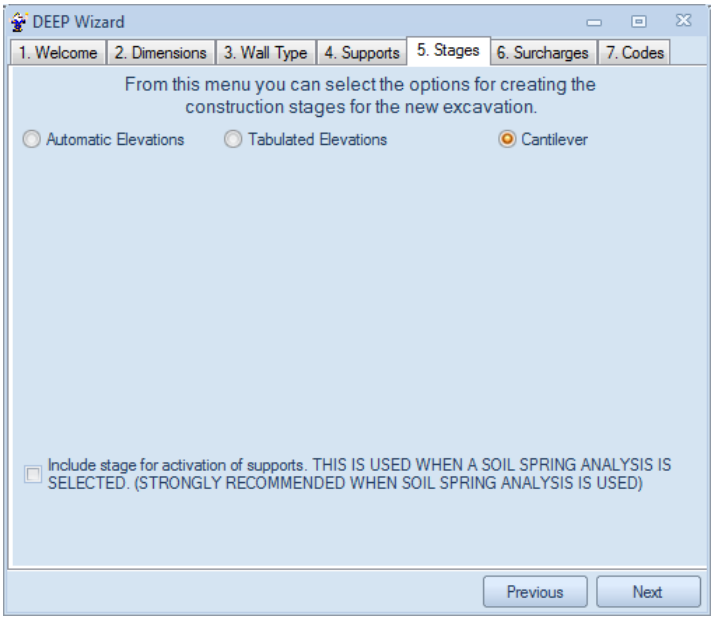

DEEP Wizard
1. Welcome
2. Dimensions
3. Wall Type
4. Supports
5. Stages
6. Surcharges
7. Codes
From this menu you can select the options for creating the construction stages for the new excavation.
Automatic Elevations
Tabulated Elevations
Cantilever
Include stage for activation of supports. THIS IS USED WHEN A SOIL SPRING ANALYSIS IS SELECTED. (STRONGLY RECOMMENDED WHEN SOIL SPRING ANALYSIS IS USED)
Previous
Next

Figure 2.4.9.c: Wizard – Stages Tab, Cantilever.

- Surcharges Tab: Here the user can define a loading surcharge that will be applied as an external load to the wall.

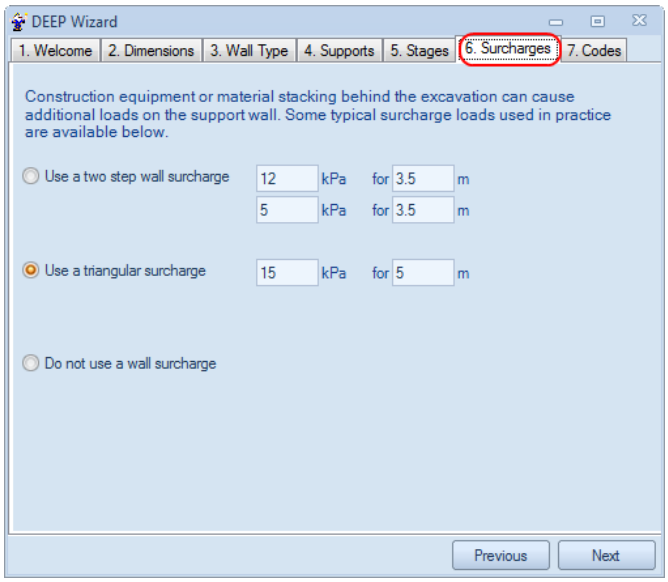


Figure 2.4.10: Wizard – Surcharges Tab.

We can choose between two step wall surcharge, triangular surcharge or not using a wall surcharge at all.

- Codes Tab: In this tab we can define the structural and geotechnical codes used to analyze this project.

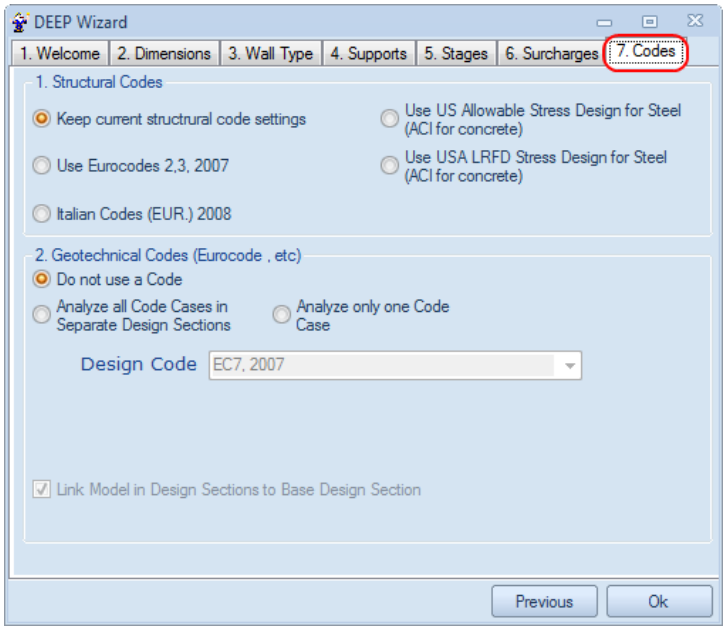

DEEP Wizard
1. Welcome
2. Dimensions
3. Wall Type
4. Supports
5. Stages
6. Surcharges
7. Codes
1. Structural Codes
Keep current structural code settings
Use US Allowable Stress Design for Steel (ACI for concrete)
Use Eurocodes 2,3, 2007
Use USA LRFD Stress Design for Steel (ACI for concrete)
Italian Codes (EUR.) 2008
2. Geotechnical Codes (Eurocode , etc)
Do not use a Code
Analyze all Code Cases in Separate Design Sections
Analyze only one Code Case
Design Code
EC7, 2007
Link Model in Design Sections to Base Design Section
Previous
Ok

Figure 2.4.11: Wizard – Codes Tab.

- **Settings:** by pressing the button

, the Default Settings dialog appears.

➢ General tab: In this tab we can define the default units, company and engineer name and the Auto save directory. The option save files in default directory saves temporary progress files in the folder where the .deep file is saved.

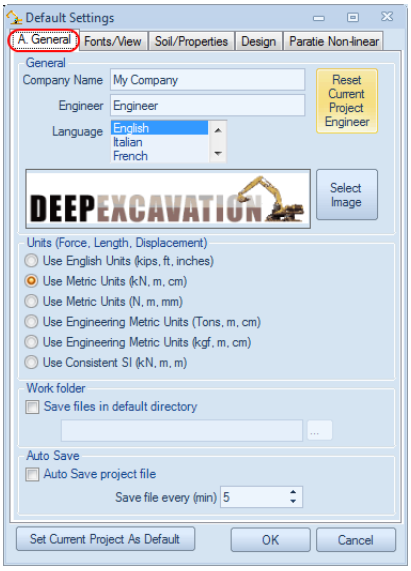


Figure 2.4.12: Settings – General Tab.

➢ Fonts/View tab: In this tab we can define the fonts and some other viewing options.

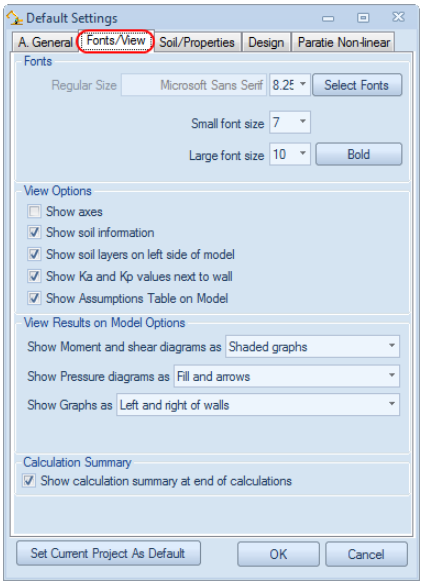


Figure 2.4.13: Settings – Fonts/View Tab.

The available view options are:

- Show axes.
- Show soil information.
- Show soil layers on left side of model.
- Show Ka and Kp values next to the wall.
- Show assumptions table on model.

Furthermore, here we can define some view options regarding the result presentation.

➢ Soil/Properties tab: Here we can define the steel member standard (European or US) and whether to read the members metric database or not.

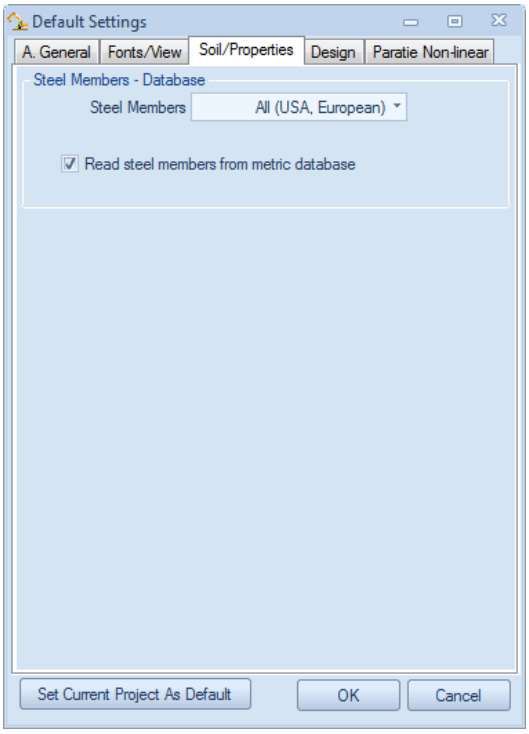


Figure 2.4.14: Settings – Soil/Properties Tab.

- Design tab: Here we can define the default code used for structural design of steel and concrete members.

Figure 2.4.15: Settings – Design Tab.

- Paratie – Non-linear tab: Here we can define the default folder location for Non-linear analysis engine. It is recommended that the initial settings are not modified.

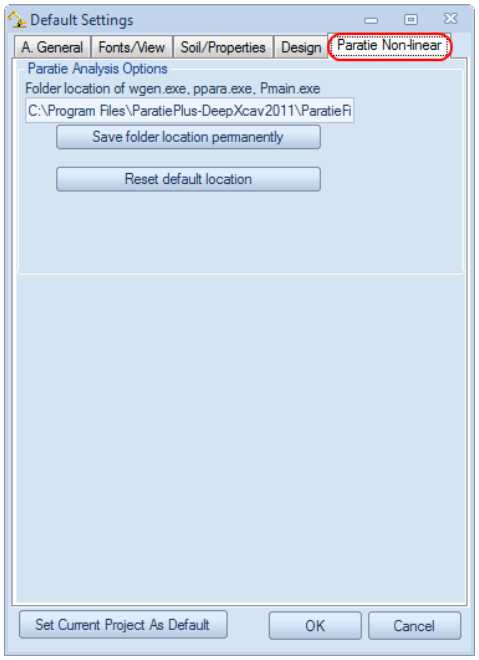


Figure 2.4.16: Settings – Paratie – Non-linear Tab.

2.5 Properties menu

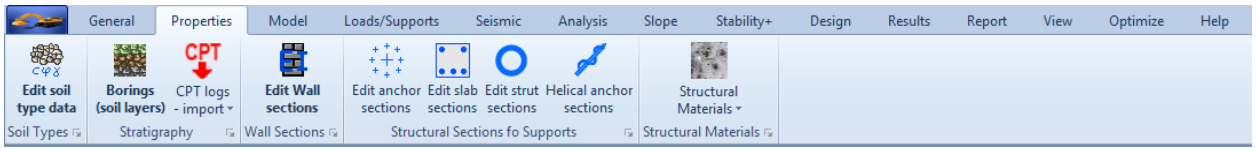


Figure 2.5.1: The Properties tab menu.

- **Edit soil type data:** by pressing the button

, the soil properties form appears. Here we can add, delete and modify available soils by changing their type, the general properties like unit weights, strength parameters and permeability, modify the elastoplastic parameters and modify the bond resistance for tiebacks. A soil can be used in a boring more than one time. A number of estimation tools that help the user estimate values are also included. **Paragraph 3.4 includes all the options that are available in this form**.

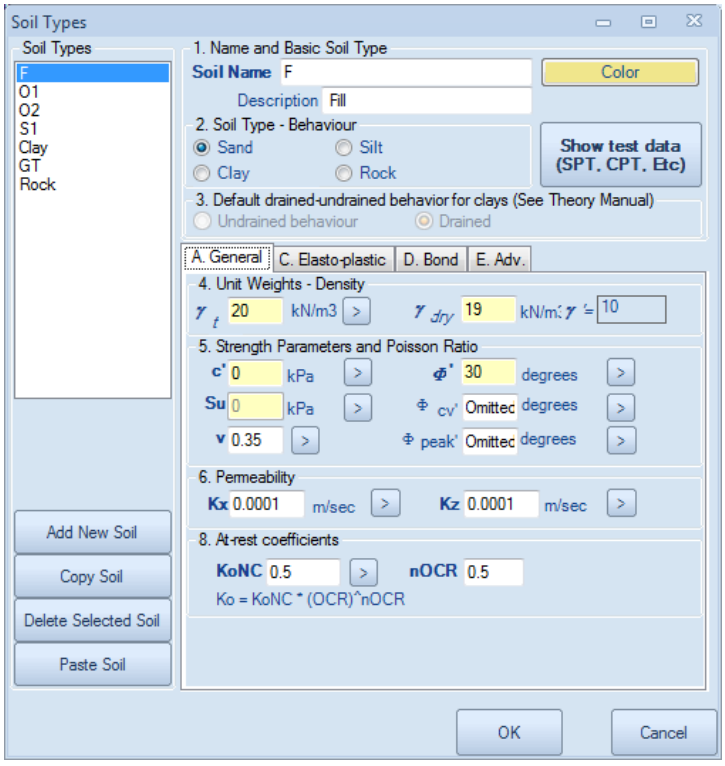


Figure 2.5.2: Edit soil properties dialog.

- **Borings (Soil layers):** by pressing the button

, the soil layer dialog appears. In this dialog we can edit the borings available for use in the project. In each boring the user can add soil layers. To do this, we can type the new soil layer's elevation, choose the soil type from the list of soil types and define the new layers OCR and Ko. In addition, by clicking on Edit button, we can modify the selected soil's properties **(see paragraph 3.4).** The coordinates X and Y refer to the plan location of the boring and do not affect analysis results.

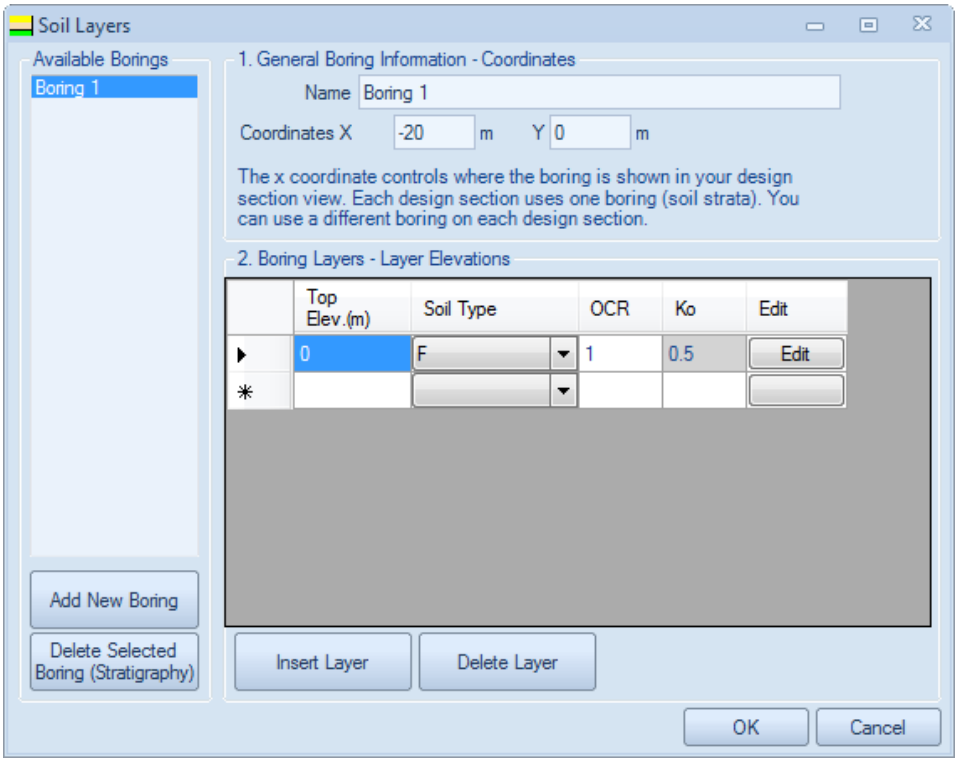
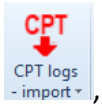


Figure 2.5.3: Edit soil layers dialog.

- **CPT logs:** by pressing the button 

, we can add borings and soils by using a CPT test results file as performed by Geologismiki CPT. The options of Figure 2.5.4.a are available. By choosing a CPT log input file, the following dialog appears:

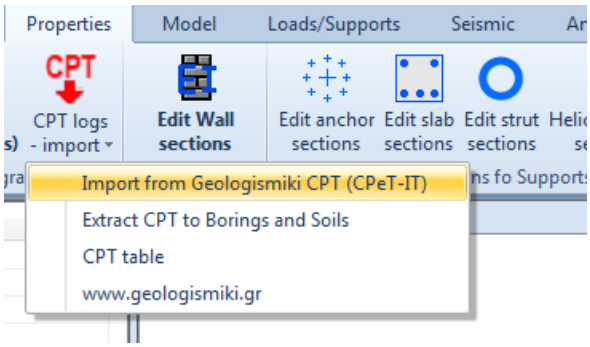


Figure 2.5.4.a: Available options for CPT logs.

The following options are available:

Import from Geologismiki CPT	Select a CPT file to import
Extract CPT to Borings and soils	Choose to add the soils from CPT log to the model's soils and borings databases
CPT table	This opens the CPT dialog
www.geologismiki.gr	This leads to the site of Geologismiki

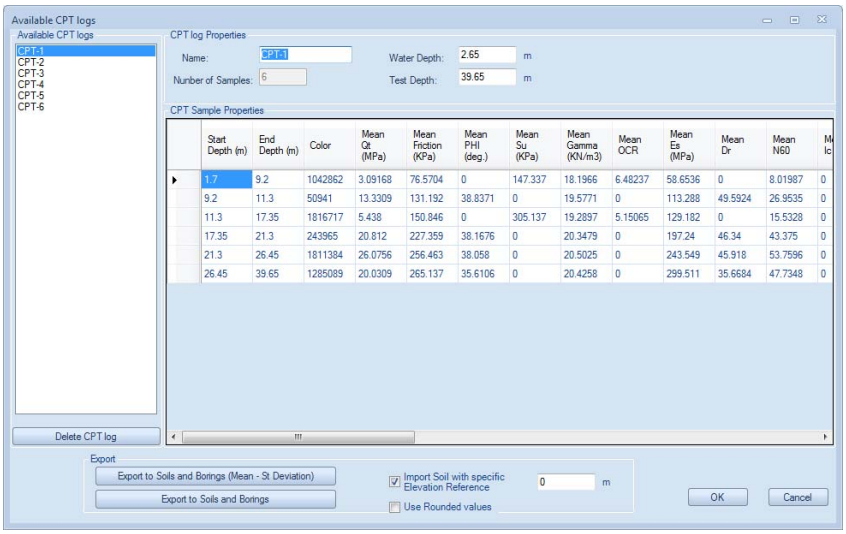


Figure 2.5.4.b: Available CPT logs dialog.

In this dialog we can export the CPT test borings and soils to the program's database.

- **Edit wall sections:** by pressing the button

, the Edit wall properties dialog appears. Here we can choose the wall type and dimensions, choose the wall sections and edit the rebar options for concrete walls. **The properties that exist on this form and can be modified are described in paragraph 3.6.**

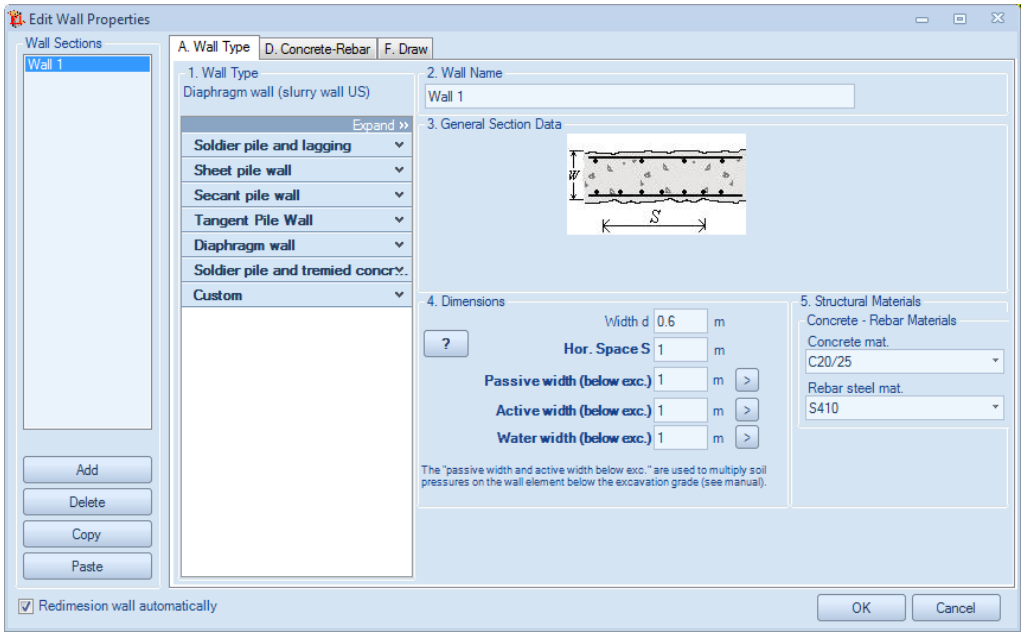
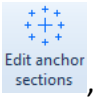


Figure 2.5.5: Edit wall properties dialog.

- **Edit anchor sections:** By pressing the button 

, we can edit the structural and geotechnical properties of the sections used in anchors and tiebacks, or select to import specific sections from the program's database. **The properties that exist on this form and can be modified are described in paragraph 3.8.**

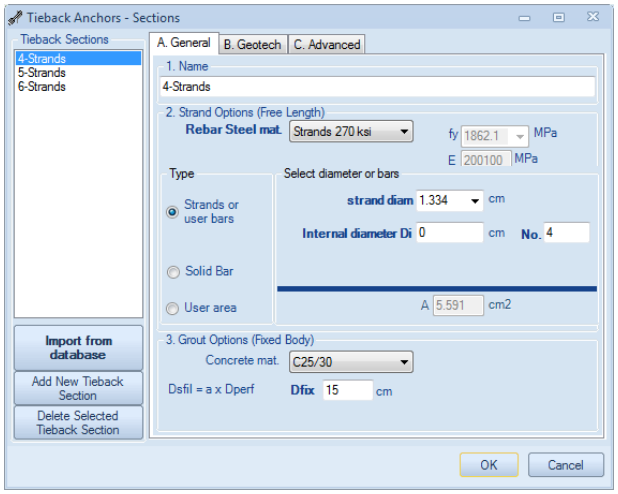
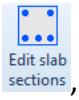


Figure 2.5.6: Edit anchor sections dialog.

- **Edit slab sections:** By pressing the button 

, we can edit the structural and reinforcement properties of the slab sections that are included in the model. **The properties that exist on this form and can be modified are described in paragraph 3.10.**

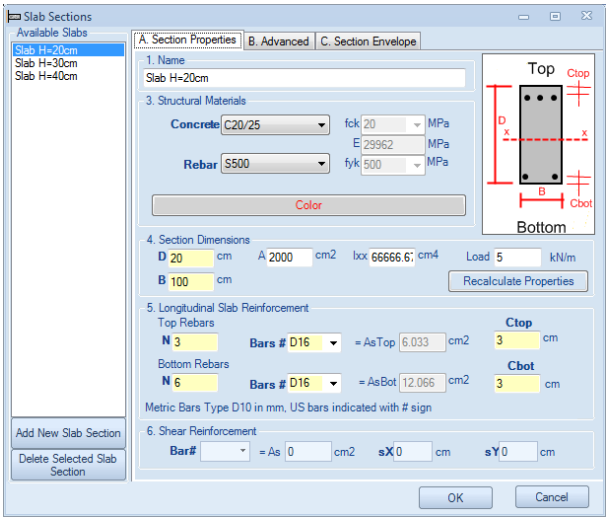


Figure 2.5.7: Edit slab sections dialog.

- **Edit strut sections:** By pressing the button

we can edit the structural properties of the strut sections that are be included in the model. **The properties that exist on this form and can be modified are described in paragraph 3.9**.

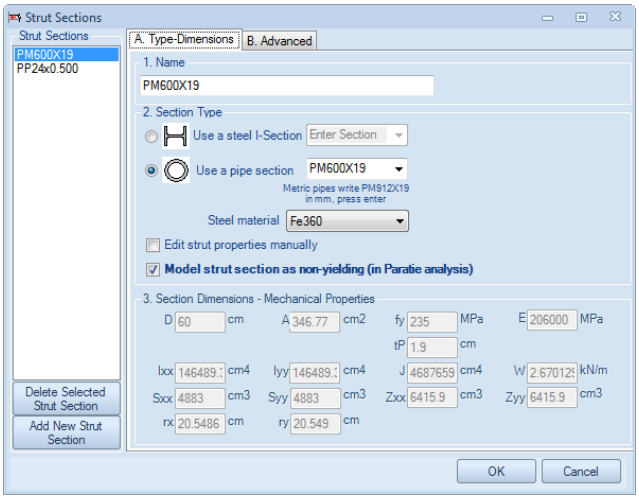
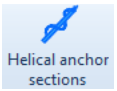


Figure 2.5.7: Edit slab sections dialog.

- **Edit helical anchor sections:** By pressing the button 

we can edit the structural and geotechnical properties of the helical anchor sections (can be used in tiebacks). **The properties that exist on this form and can be modified are described in paragraph 3.8**.

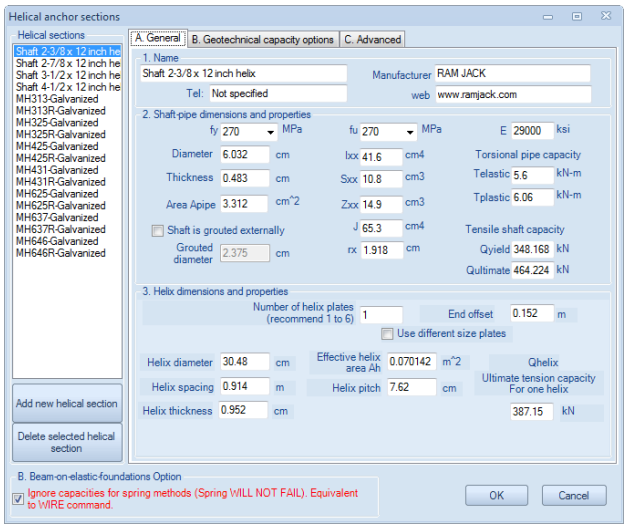


Figure 2.5.8: Edit helical anchor sections dialog.

- **Structural materials:** In this area we can edit the structural material properties.

  The following options are available (Figure 2.5.9):

Concrete
Reinforced steel
Steel
Timber (wood)
User defined material

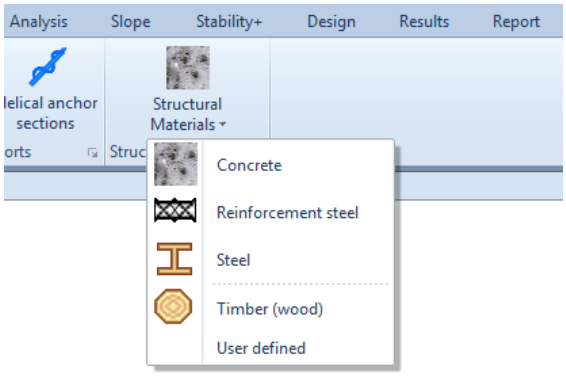


Figure 2.5.9: Structural material options.

- **Edit steel properties:** By pressing the button we can edit the structural steel properties. We can import already available materials from the "Import standard steel materials" box.

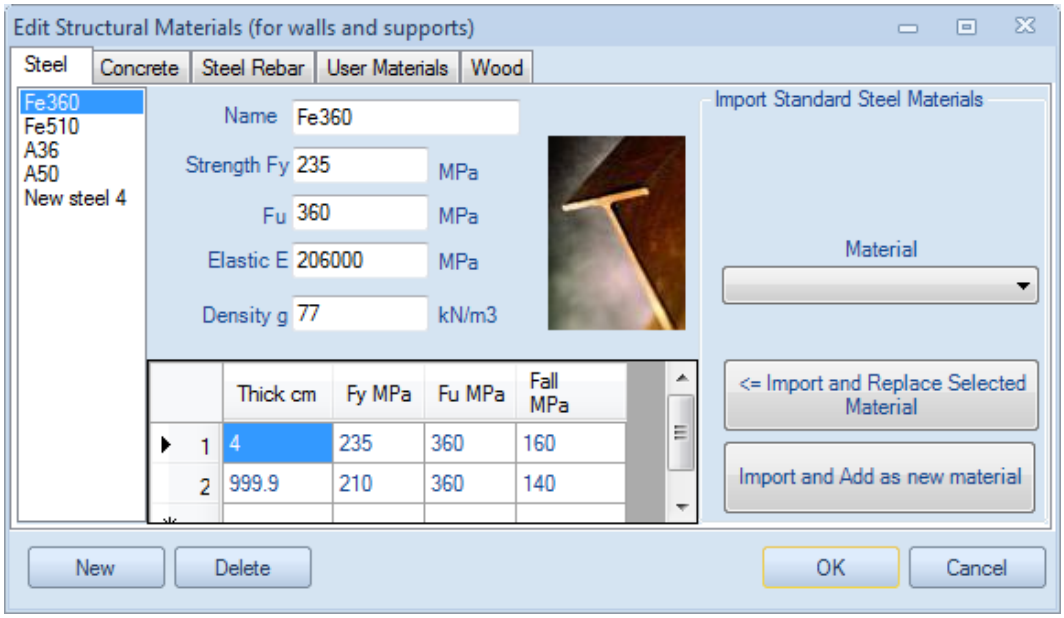


Figure 2.5.10: Edit structural steel properties dialog.

In this form we can define the following properties:

The steel name
The yield strength Fy
The ultimate strength Fu
The modulus of elasticity E
The density g
The steel material used
Import and replace selected material
Import and add as a new material

- **Edit concrete properties:** By pressing the button we can edit the concrete properties. We can import already available materials from the "Import standard concrete materials" box.

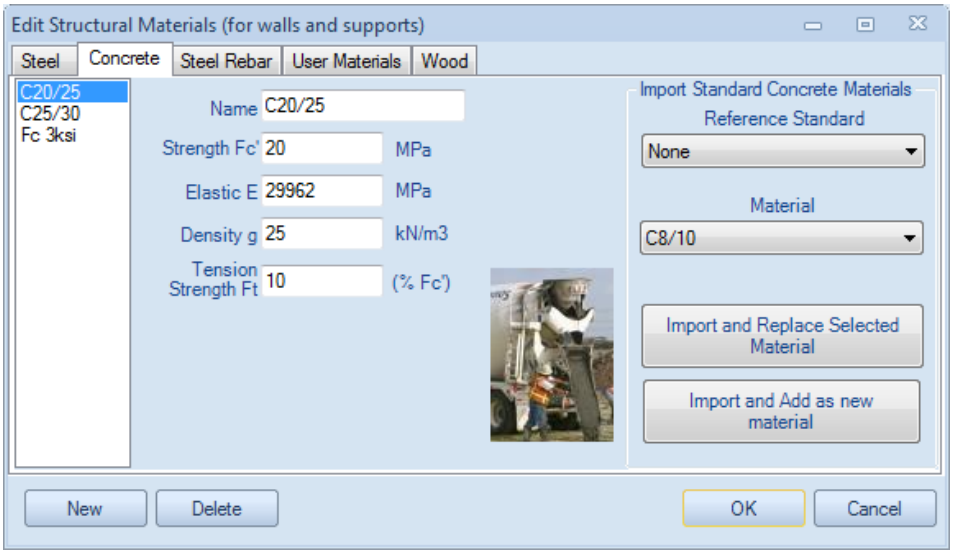


Figure 2.5.11: Edit concrete properties dialog.

In this form we can define the following properties:

The steel name
The concrete strength Fc
The tension strength Ft (% of compressive strength)
The modulus of elasticity E
The density g
The standard concrete material reference standard
The concrete material
Import and replace selected material
Import and add as a new material

- **Edit reinforcement steel properties:** By pressing the button we can edit the reinforcement (rebar) steel properties. We can import already available materials from the "Import standard rebar materials" box. Steel reinforcement materials can be used for tiebacks and reinforced concrete sections.

Figure 2.5.10: Edit reinforcement steel properties dialog.

In this form we can define the following properties:

The steel name
The yield strength Fy
The modulus of elasticity E
The standard rebar material reference standard
The steel material used
Import and replace selected material
Import and add as a new material

- **Edit user material properties:** By pressing the button User defined we can create a custom material, by defining the modulus of elasticity. User materials are used for custom defined wall sections.

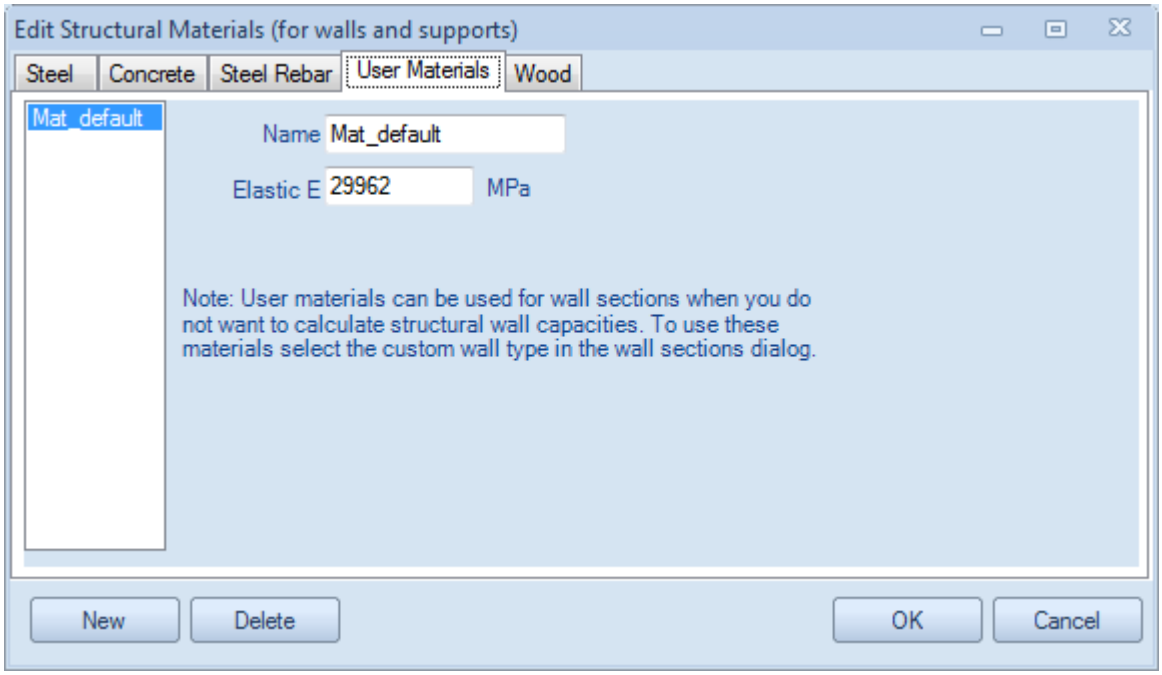


Figure 2.5.11: Edit user material properties dialog.

- **Edit wood properties:** By pressing the button we can modify the wood material properties. Wood (timber) materials are used in laggings for soldier pile walls.

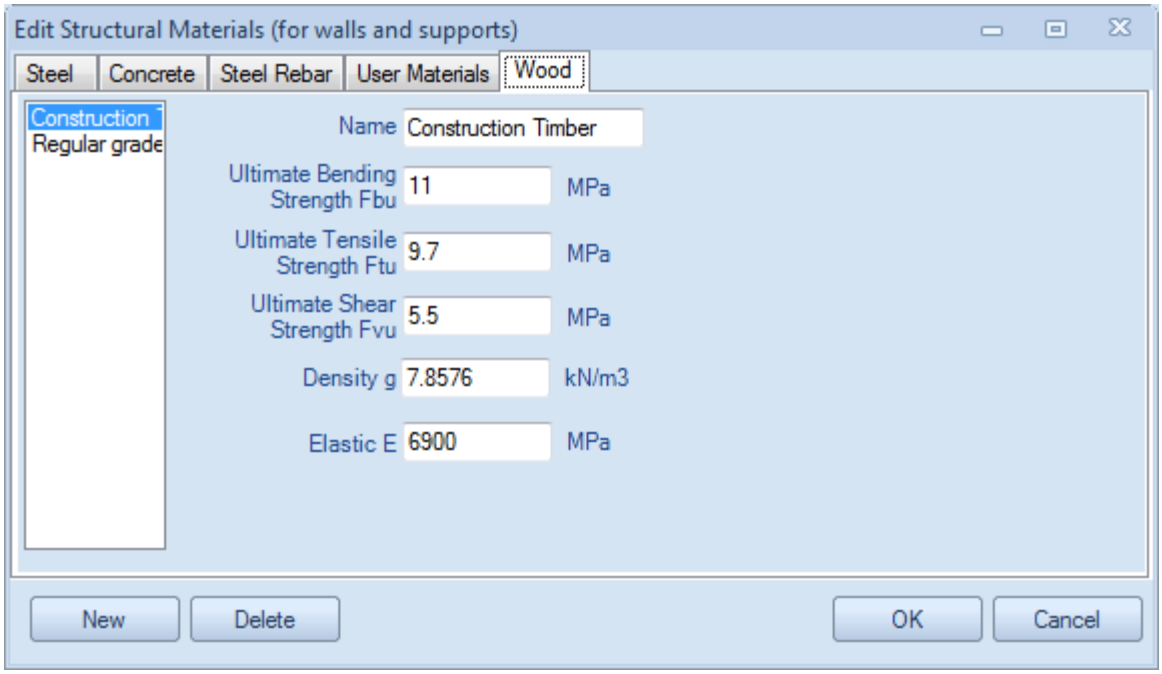


Figure 2.5.12: Edit wood properties dialog.

In this form we can define the following properties:

The wood name
The ultimate bending strength Fbu
The ultimate tensile strength Ftu
The ultimate shear strength Fvu
The modulus of elasticity E
The density g

2.6 Model menu

Figure 2.6.1: The Model tab menu.

- **Custom layers:** by pressing the button Custom layers, we can choose to use the DeepXcav's Custom layer mode and use non parallel soil layers. In this mode, a model will not use a borehole to define soil layers. Instead, the user can define arbitrary layer lines from left to right and create non-horizontal soil layer conditions.

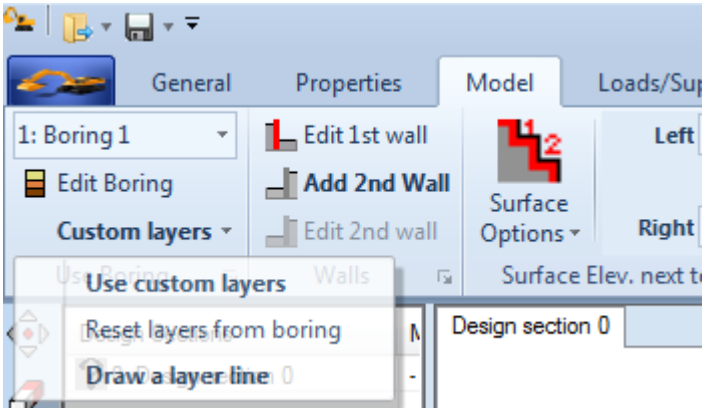
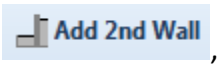


Figure 2.6.2: Custom layers mode.

- The option Use custom layers switches on the custom layer mode.
- The reset layers from boring option, resets all custom layer lines to horizontal using the boring applicable for the current design section.
- The draw layer line tool enables the user to draw a layer line by clicking in the model (left to right). Press enter to complete.

- **Add second wall:** by pressing the button 

, we can add a second wall to the model on the right side of the existing wall.

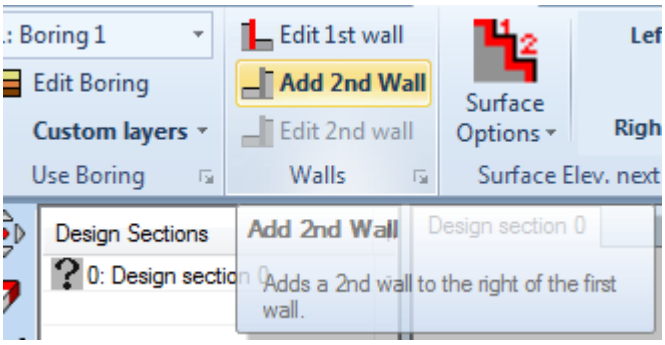
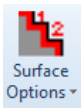


Figure 2.6.3: Add second wall.

- **Surface options:** By clicking on the button 

we can edit the surface options. These options are presented in the table below. In addition, we can change the elevation next to the wall in the area signed in Figure 2.6.4.

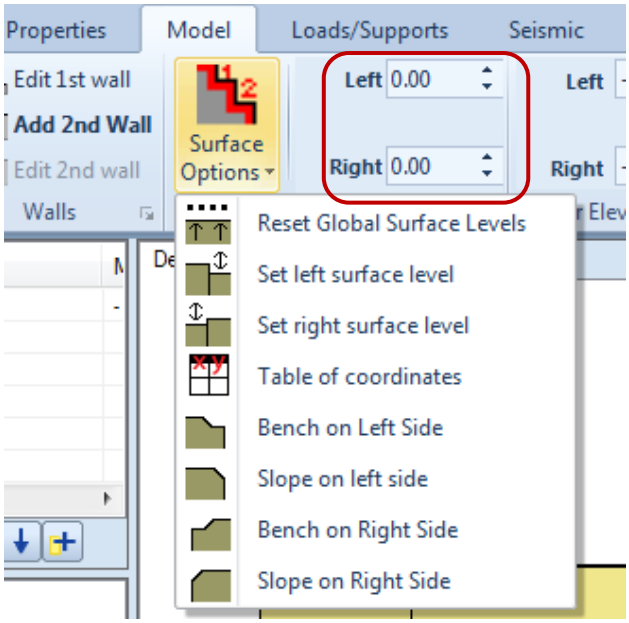


Figure 2.6.4: Surface options.

Icon	Description
	Resets global surface levels
	Sets right surface level
	Sets left surface level
	Table of coordinates of surface points
	Create a bench on the left side
	Create a slope on the left side
	Create a bench on the right side
	Create a slope on the right side

- **Water elevation:** In this area we can define the water elevations next to the wall.

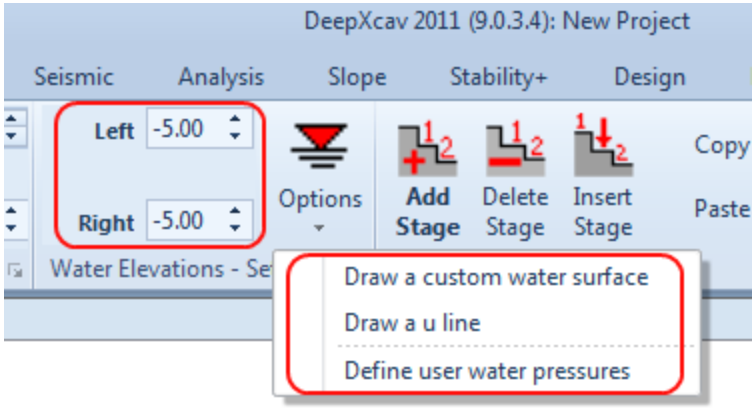


Figure 2.6.5: Water elevation - settings.

- With the draw custom water surface tool, we can draw a non-horizontal groundwater table. To do this, select this option and then start clicking the left mouse button from left to right (press enter to complete). To delete the custom water line, move on top in the model and press delete.
- The draw a U line tool is used to draw a line of constant pore pressure in the model. This line is only used in slope stability analysis.

- The define user water pressures tool launches the dialog for defining custom values of water pressures next to the walls. Please note that in the non-linear engine, two consecutive zero values of water pressure still count in increasing the total vertical stress by γ_w (see theory manual).

By clicking on the button , the Ground water table dialog shows up.

Figure 2.6.6: Ground water table.

The following table presents the options that are included in the ground water table dialog.

Define the retained side water elevation (left side)
Option to use hydrostatic ground water pressures
Option to use simplified flow net for ground water (1D flow calculations)
Option to perform full flow net analysis (finite difference)
Choice for constant head on lower boundary (if the previous option is selected)
Option to use user defined water pressures
Option to maintain center region water at subgrade
Option to dewater center region water below subgrade
Option to use general elevation for center region water
Choice to create seal at excavation bottom
Define the water density γ_w
Choice to specify different center water elevation for second wall (this option is available only if a second wall is used in the model)

- **Stages:** In this area we can add, delete, insert or copy a construction stage.

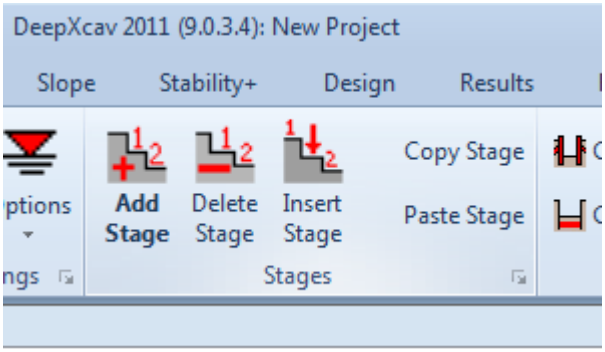


Figure 2.6.7: Water elevation - settings.

Icon	Description
	Add a new construction stage
	Deletes the current construction stage
	Insert a construction stage after the current stage
Copy Stage	Copy selected construction stage
Paste Stage	Paste construction stage

- **Advanced:** In this area we can modify structural material and soil properties during a specific stage. A change property command is valid for all subsequent stages, unless it is superseded by another change command or another model change. Please use these options with care.

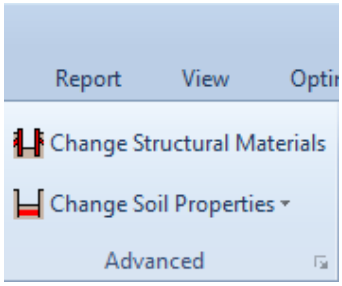


Figure 2.6.8: Advanced options.

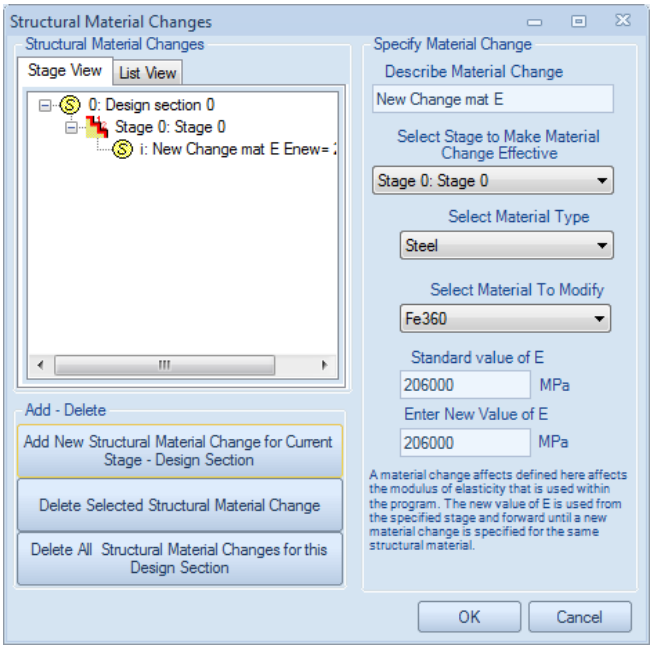

Structural Material Changes
Structural Material Changes
Stage View
List View
0: Design section 0
Stage 0: Stage 0
i: New Change mat E Enew=
Add - Delete
Add New Structural Material Change for Current Stage - Design Section
Delete Selected Structural Material Change
Delete All Structural Material Changes for this Design Section
Specify Material Change
Describe Material Change
New Change mat E
Select Stage to Make Material Change Effective
Stage 0: Stage 0
Select Material Type
Steel
Select Material To Modify
Fe360
Standard value of E
206000
MPa
Enter New Value of E
206000
MPa
A material change affects defined here affects the modulus of elasticity that is used within the program. The new value of E is used from the specified stage and forward until a new material change is specified for the same structural material.
OK
Cancel

Figure 2.6.9: Change structural materials dialog.

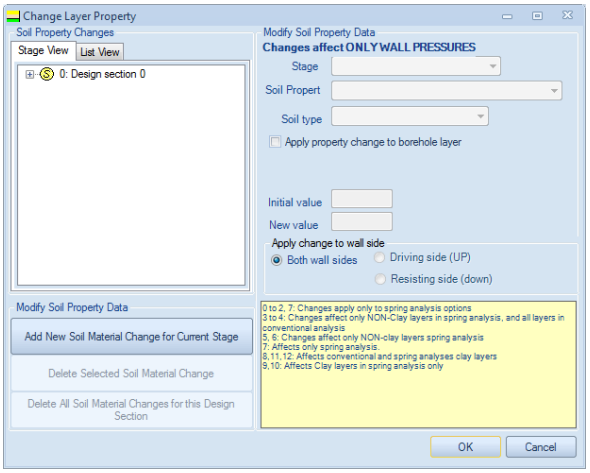


Figure 2.6.10: Change soil properties dialog.

In the Change soil properties dialog, the following properties are available:

Describe material change
Select stage to Make material change effective
Select material type
Select material to modify
Standard value of modulus of elasticity E
New value of modulus of elasticity E

- **Draw left wall beam:** By pressing the button

, we can draw a second wall beam additional to the existing wall and modify its properties, as shown in Figures 2.6.11 and 2.6.12. To draw a wall beam click on one point near the wall and then select the next wall beam point.

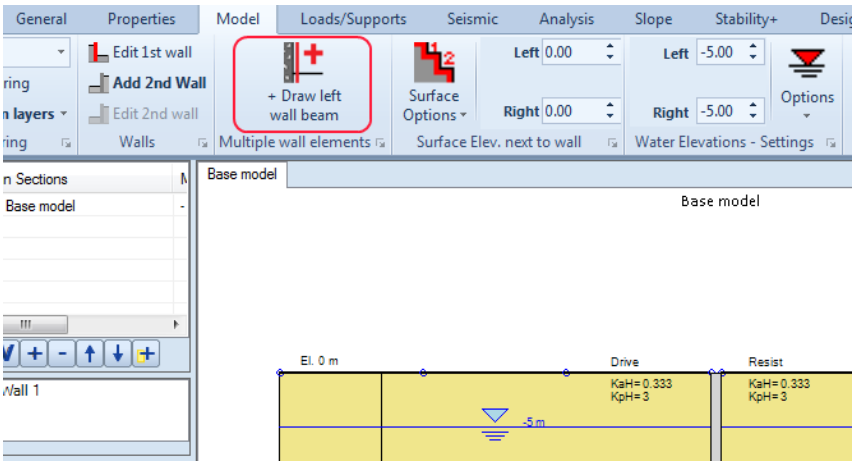


Figure 2.6.11: The draw left wall beam button.

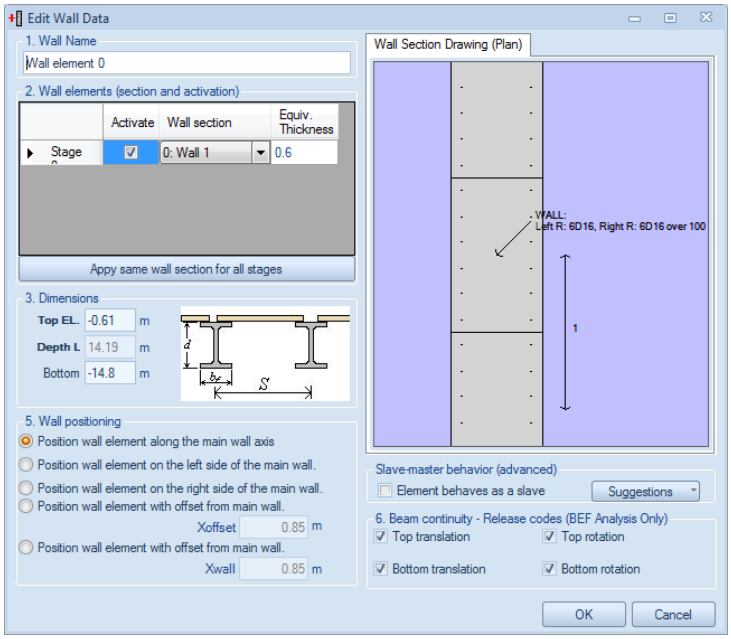


Figure 2.6.12: Edit wall beam data dialog.

The options available in the Edit wall data form are described in paragraph 3. 7. The option Draw right wall beam is available when we choose to use a second wall in the model (Figure 2.6.13).

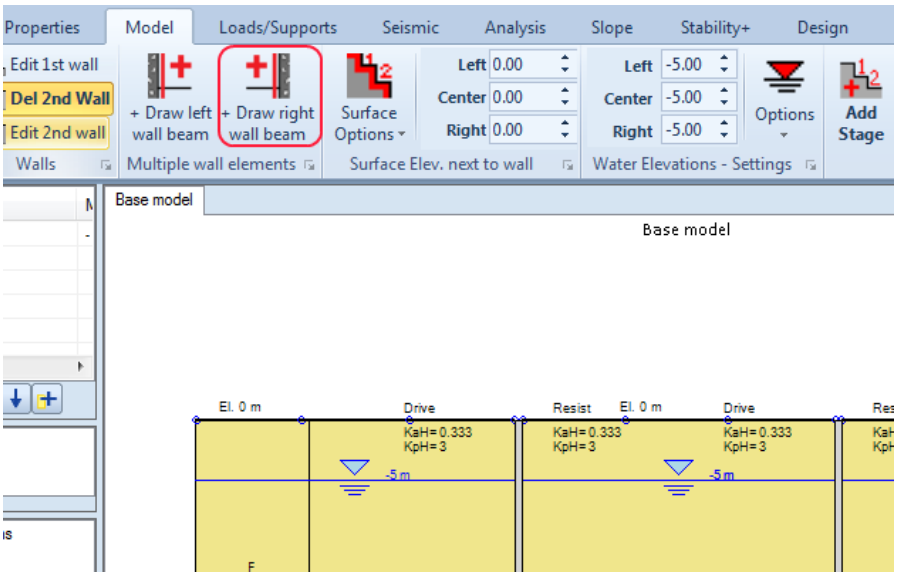


Figure 2.6.13: The draw right wall beam button.

2.7 Load/Support menu

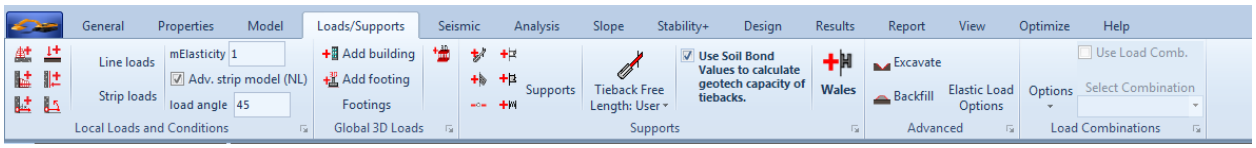


Figure 2.7.1: The Load/Support tab menu.

Tool	Description
	Adds a tieback – anchor (first click on the wall and then in the ground).
	Adds a strut (first click on the wall and then in the ground, or on the second wall).
	Adds a raker click on the wall and then in the ground).
	Adds a slab (first click on the wall and then define the end point of the slab).
	Adds a fixed support (click on the wall to add a fixed support).
	Adds a spring (click on the wall to add a spring).
	Adds a surface surcharge (define the start and end point of the surcharge).
	Adds a surface line load (click a surface point to add a point load).
	Adds a surcharge on the wall (define two wall points to add a surcharge).
	Adds a line load on the wall (define a wall point to add a wall point load)
	Adds a prescribed condition at a wall (click on the wall to add a prescribed condition). A prescribed condition is a predefined displacement or wall rotation (non-linear analysis)
	Adds a footing load (3D) (define a point where to install a footing load).
	Creates a new building (define a point where to install a building).
	Performs an excavation (click on screen to define the excavation limits).
	Performs a backfill operation (click on screen to define the backfill limits).

- **Add building:** By pressing the button Add building , we can add a building in the model, using the building wizard dialog.
  The properties contained in this dialog are described in paragraph 4.7.

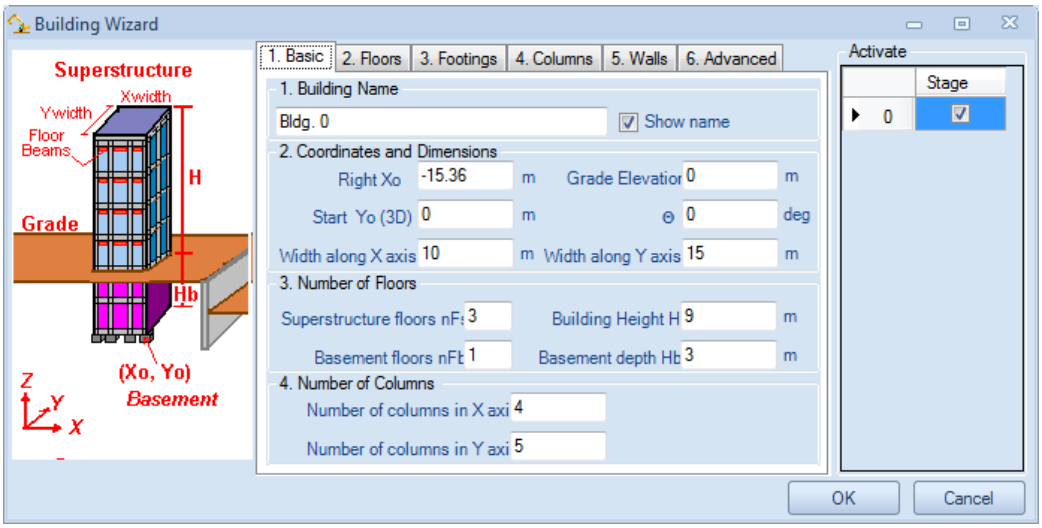


Figure 2.7.2: Building Wizard dialog.

- **Add footing:** By pressing the button Add footing , we can add a footing load in the model, using the footing options dialog.
  The properties contained in this dialog are described in paragraph 4.6.

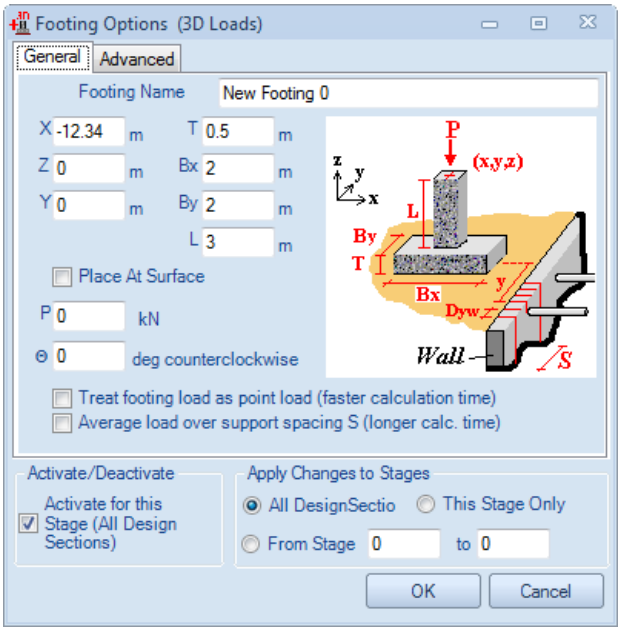


Figure 2.7.3: Footing options dialog.

- **Wales:** By pressing the button

we can add a wale beam to a support. The wale beam dialog opens and the user can define the type and the loading case of the waler. **The properties contained in this dialog are described in paragraph 3.14.**

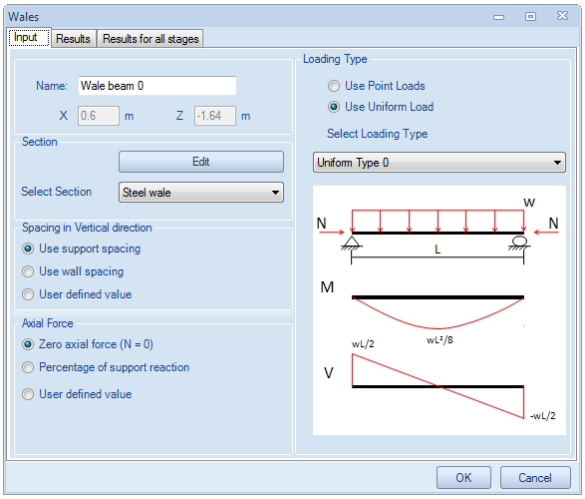


Figure 2.7.4: Wale beam dialog.

- **Load combinations:** By pressing the button

, the Load combinations dialog appears (Figure 2,7,5). In this dialog we can create load combinations. Each load combination can control how the load is treated when a design approach is applied, in terms of the load character (favorable, unfavorable, automatic, ignored). This definition can be applied on each load that has been applied to the model (surcharges, line loads, footings, buildings and other 3D loads).

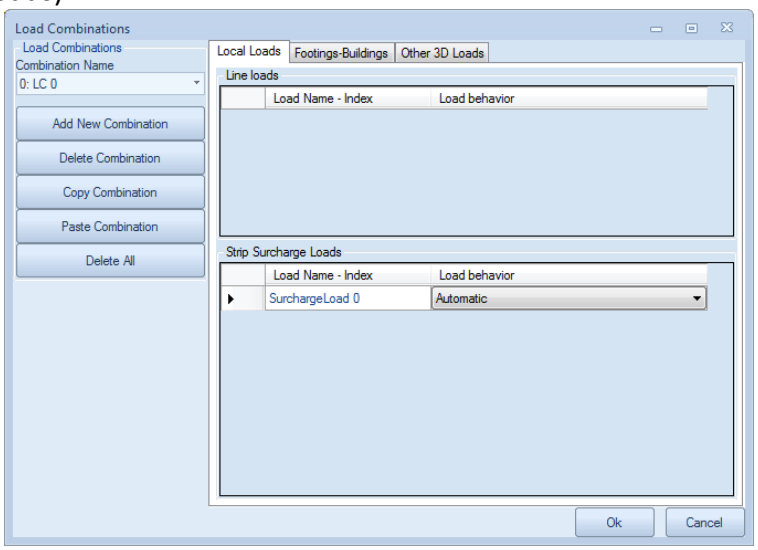


Figure 2.7.5: Load combinations dialog – Local loads.

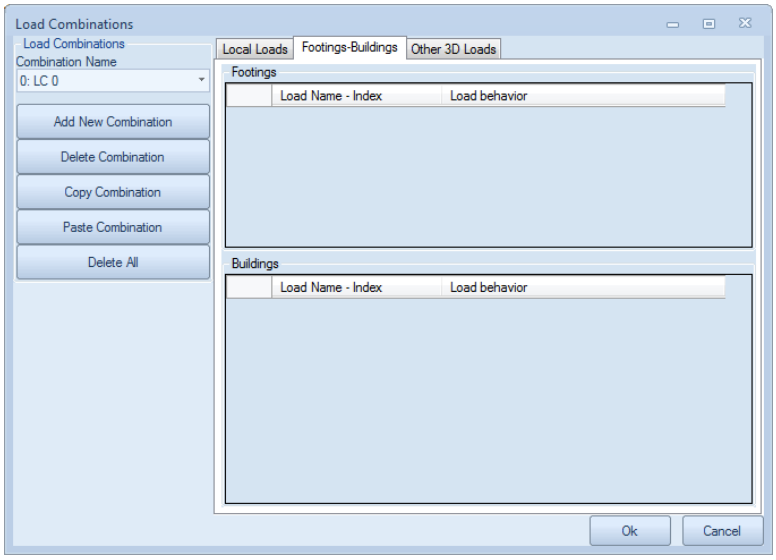


Figure 2.7.6: Load combinations dialog – Footings-Buildings.

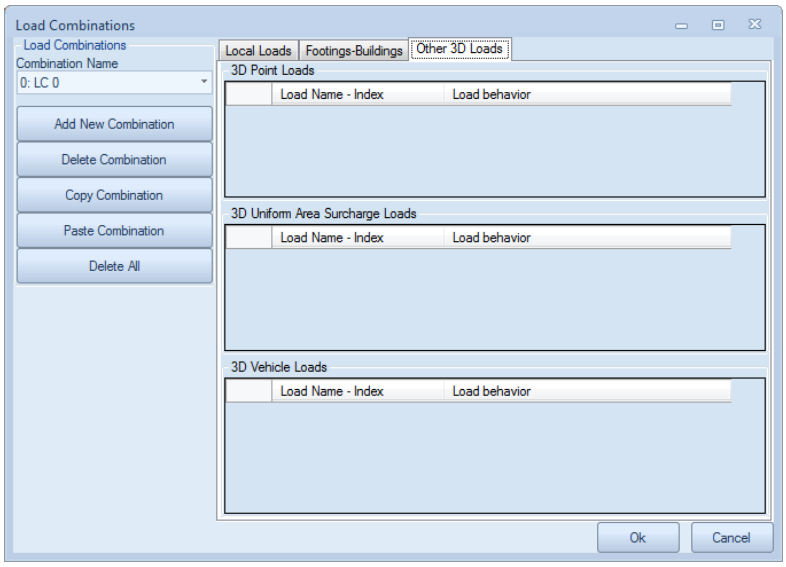


Figure 2.7.7: Load combinations dialog – Other 3D loads.

2.8 Seismic menu

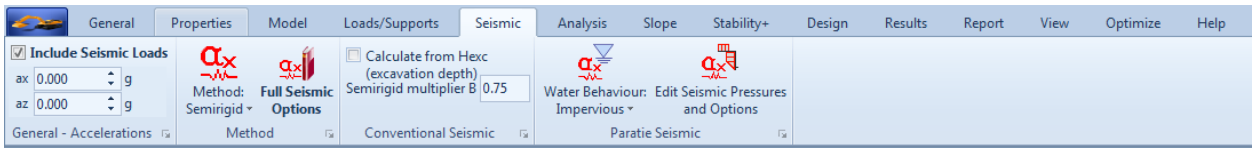


Figure 2.8.1: The Seismic tab menu.

In this menu, we can choose to include Seismic options and loads in the model's analysis. The following options are included:

- **General – accelerations:** In this area we can define the earthquake's accelerations for both directions (horizontal ax and vertical az).

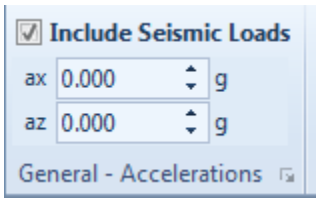


Figure 2.8.2: Seismic accelerations.

- **Method:** In this area we can choose the Seismic method. We can either choose from the available list (Figure 2.8.3) or press the full seismic options button and load the seismic effects dialog (Figure 2.8.4).

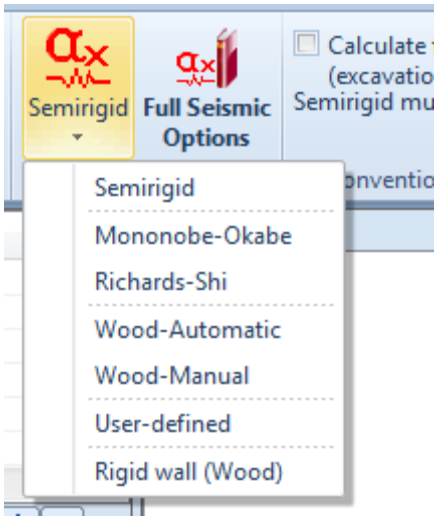


Figure 2.8.3: Seismic accelerations.

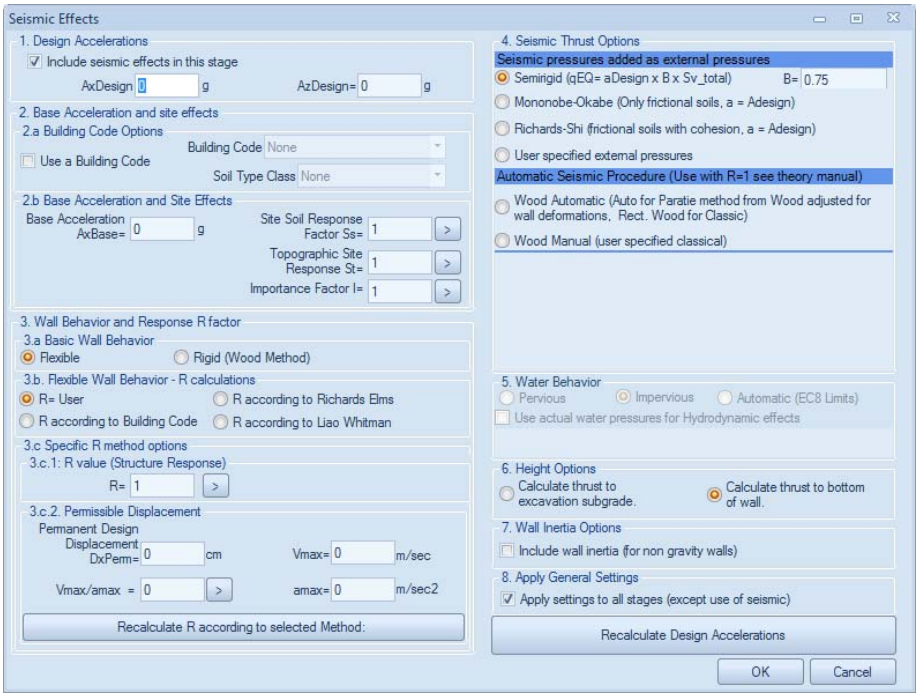

Seismic Effects
1. Design Accelerations
Include seismic effects in this stage
AxDesign 0 g
AzDesign= 0 g
2. Base Acceleration and site effects
2.a Building Code Options
Use a Building Code
Building Code None
Soil Type Class None
2.b Base Acceleration and Site Effects
Base Acceleration AxBase= 0 g
Site Soil Response Factor Ss= 1
Topographic Site Response St= 1
Importance Factor I= 1
3. Wall Behavior and Response R factor
3.a Basic Wall Behavior
Flexible
Rigid (Wood Method)
3.b. Flexible Wall Behavior - R calculations
R= User
R according to Richards Elms
R according to Building Code
R according to Liao Whitman
3.c Specific R method options
3.c.1: R value (Structure Response)
R= 1
3.c.2. Permissible Displacement
Permanent Design Displacement DxPerm= 0 cm
Vmax= 0 m/sec
Vmax/amax = 0
amax= 0 m/sec2
Recalculate R according to selected Method:
4. Seismic Thrust Options
Seismic pressures added as external pressures
Semirigid (qEQ= aDesign x B x Sv_total)
B= 0.75
Mononobe-Okabe (Only frictional soils, a = Adesign)
Richards-Shi (frictional soils with cohesion, a = Adesign)
User specified external pressures
Automatic Seismic Procedure (Use with R=1 see theory manual)
Wood Automatic (Auto for Paratie method from Wood adjusted for wall deformations, Rect. Wood for Classic)
Wood Manual (user specified classical)
5. Water Behavior
Pervious
Impervious
Automatic (EC8 Limits)
Use actual water pressures for Hydrodynamic effects
6. Height Options
Calculate thrust to excavation subgrade.
Calculate thrust to bottom of wall.
7. Wall Inertia Options
Include wall inertia (for non gravity walls)
8. Apply General Settings
Apply settings to all stages (except use of seismic)
Recalculate Design Accelerations
OK
Cancel

Figure 2.8.4: Seismic effects dialog.

The following=g table presents the options that are available in the Seismic effects dialog.

Choice to include seismic effects in this stage
Horizontal design acceleration Axdesign
Vertical design acceleration Azdesign
Option to use a building code
Define building code and soil type class (if the previous option is selected)
Define the horizontal base acceleration Axbase
Define the site soil response factor Ss
Define the topographic site response St
Define the Importance factor I
Option to use rigid wall behavior (wood method)
Option to use flexible wall behavior
Option to use a user defined response factor R (if the flexible wall option is selected)
Option define response factor R according to Building code (if the flexible wall option is selected)
Option define response factor R according to Richards Elms (if the flexible wall option is selected)
Option define response factor R according to Liao Whitman (if the flexible wall option is selected)
Define value of R (if user defined R option is selected)
Define the Permanent design displacement Dxperm (after the seismic event)
Define the Vmax (peak velocity)

Define the amax (peak acceleration)
Define the Vmax/amax (option > available for estimation)
Option to use Semirigid method for seismic pressures
Option to use Mononobe-Okabe method for seismic pressures
Option to use Richards-Shi method for seismic pressures
Option to use user specified external pressures
Option to use Wood automatic procedure
Option to use Wood manual procedure
Define the user defined seismic pressures for top and bottom
Define the water behavior as Pervious, impervious or automatic
Choice to use actual water pressures for Hydrodynamic effects (instead of γ_w x h_w)
Option to calculate thrust to excavation subgrade
Option to calculate thrust to bottom of the wall
Choice to include wall inertia for non gravity walls
Choice to apply settings to all stages
Recalculate design accelerations

- **Conventional Seismic:** In this area we can control options for controlling the height of application for seismic loads, and also define the Semirigid seismic multiplier B (factor for creating a rectangular seismic thrust diagram).

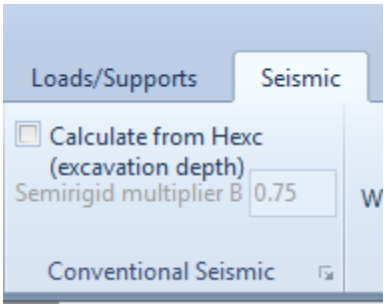


Figure 2.8.5: Conventional seismic options.

- **Paratie Seismic:** In this area we can define the water behavior (Figure 2.8.6) and seismic pressures (Figure 2.8.7) regarding the Non-linear (Paratie) analysis. The pervious and impervious options control how the water is treated in a seismic condition. In the pervious case hydrodynamic loading is applied according to the Westeergard solution. If the automatic EC8 limit option is selected, then the pervious/impervious state is automatically controlled from the horizontal soil permeability according to the limits specified in Eurocode 8.

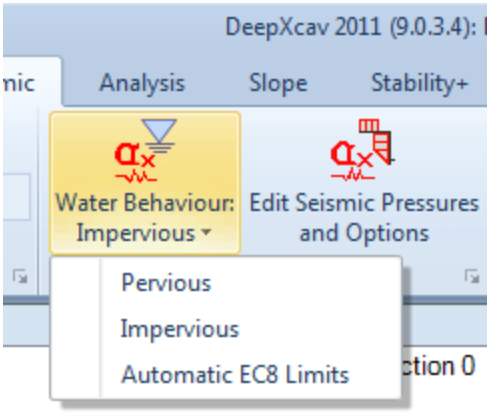


Figure 2.8.6: Water behavior options.

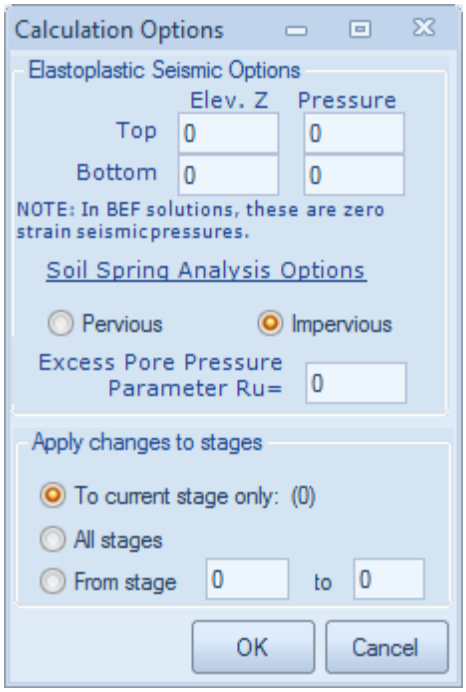


Figure 2.8.7: Elastoplastic seismic pressure options.

2.9 Analysis menu

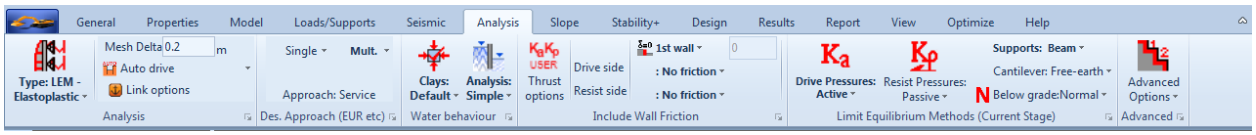


Figure 2.9.1: The Analysis tab menu.

In this menu we can edit the basic analysis settings.

- **Analysis:** In this area we can define the analysis method. We can choose to perform a conventional analysis (limit equilibrium), a non-linear analysis or a combination of both two. In addition, here we can define the spacing among the points of the wall (mesh delta). In general it is recommended that a mesh that generates at least 100 nodes is adopted.

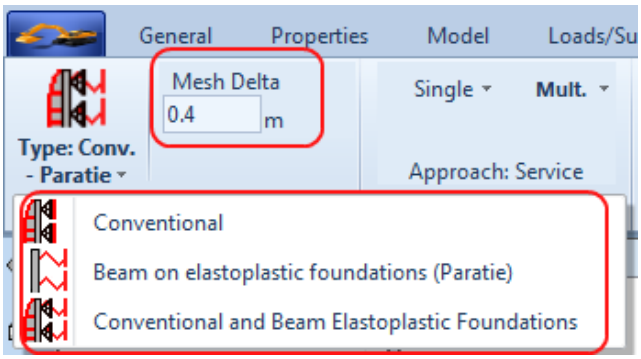


Figure 2.9.2: Analysis options.

- **Design approach:** In this area we can define the design load cases of standards. There is a possibility to choose a specific case of a design standard (Figure 2.9.3), or to generate all load cases/design approaches of a design code (Figure 2.9.4). **Paragraph 3.2 presents all these options.**

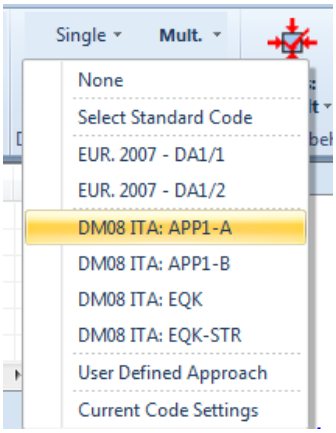


Figure 2.9.3: Single load case.

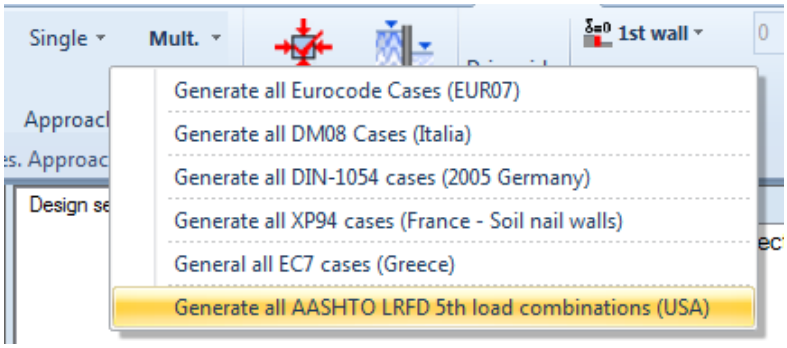

Single
Mult.
1st wall
Generate all Eurocode Cases (EUR07)
Generate all DM08 Cases (Italia)
Generate all DIN-1054 cases (2005 Germany)
Generate all XP94 cases (France - Soil nail walls)
General all EC7 cases (Greece)
Generate all AASHTO LRFD 5th load combinations (USA)

Figure 2.9.4: Multiple load cases.

- **Water behavior:** In this area we can define the clay behavior and the water analysis type for each stage (Figures 2.9.5 and 2.9.6 respectively).

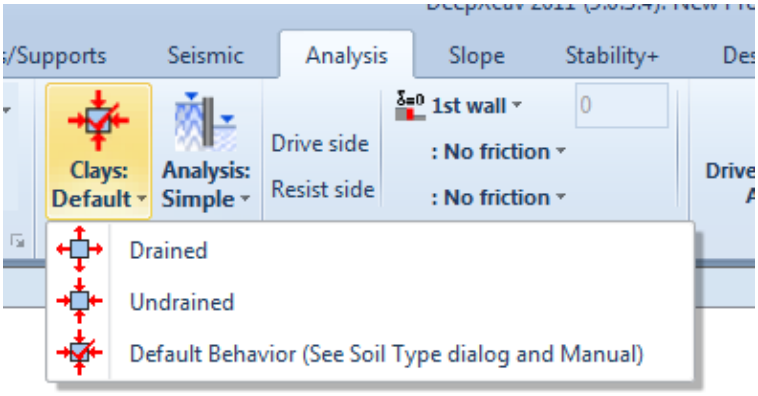


Figure 2.9.5: Clay behavior.

For drained analysis the program will use effective strength parameters (for clays c' is used in LEM analysis and global stability). The default behavior uses the initial drained/undrained setting as defined in the soil's dialog.

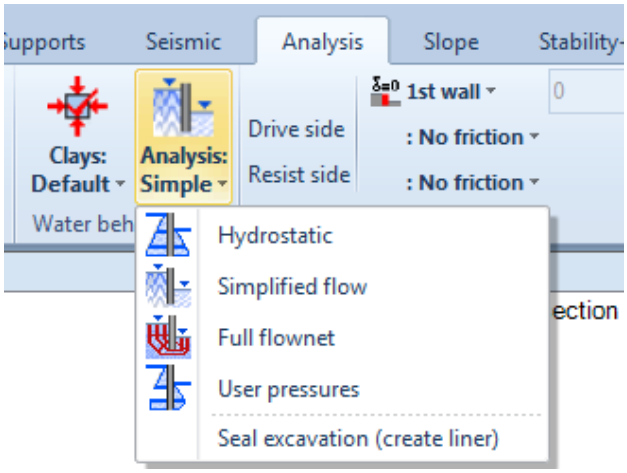
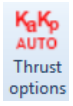


Figure 2.9.6: Water analysis options.

Hydrostatic	Water pressures are calculated as hydrostatic (γ_w x h)
Simplified flow	Water pressures next to the walls are calculated from a 1D flow analysis
Full flownet	Water pressures are calculated from a 2D finite difference flownet analysis
User pressures	Switches water pressures to the user defined values
Seal excavation	This is an advanced option. With this, we can seal the excavation and define an internal water level that is higher than the excavation level. In this case it is important to maintain positive effective vertical stress within the excavation

- **Thrust options:** By clicking on the button 

, the Ka Kp main options dialog appear (Figure 2.9.8). In this dialog, we can choose the procedure for calculating lateral earth pressure coefficients. In the most cases, the full automatic procedure will deliver the best results.

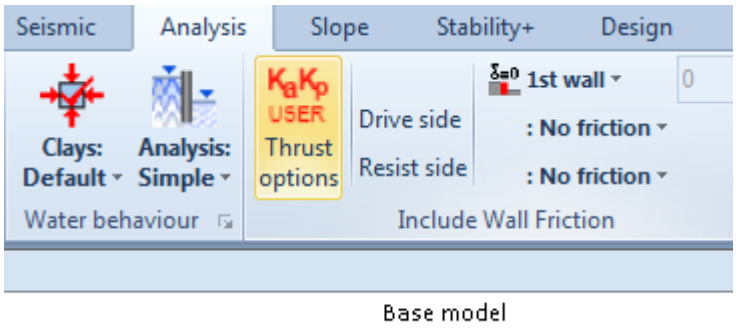


Figure 2.9.7: The thrust options button.

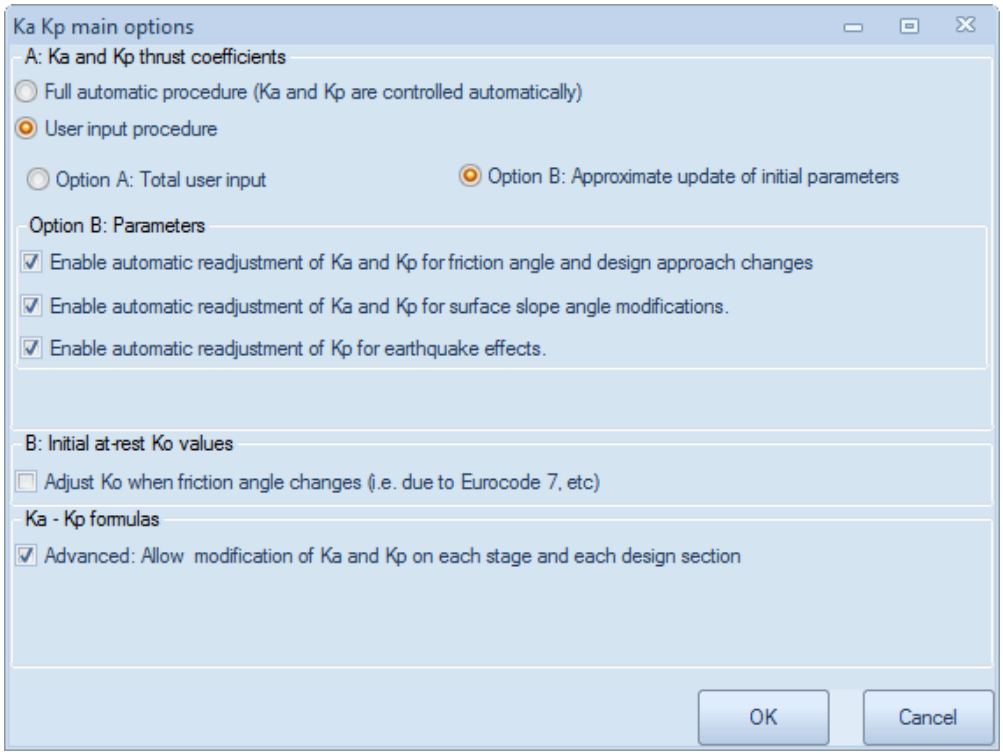

Ka Kp main options
A: Ka and Kp thrust coefficients
Full automatic procedure (Ka and Kp are controlled automatically)
User input procedure
Option A: Total user input
Option B: Approximate update of initial parameters
Option B: Parameters
Enable automatic readjustment of Ka and Kp for friction angle and design approach changes
Enable automatic readjustment of Ka and Kp for surface slope angle modifications.
Enable automatic readjustment of Kp for earthquake effects.
B: Initial at-rest Ko values
Adjust Ko when friction angle changes (i.e. due to Eurocode 7, etc)
Ka - Kp formulas
Advanced: Allow modification of Ka and Kp on each stage and each design section
OK
Cancel

Figure 2.9.8: The Ka Kp main options dialog.

In this dialog the following options are available:

Full automatic procedure (automatic selection of Ka and Kp)
User input procedure
Option A: Total user input (if the user input procedure option is selected)
Option B : Approximate update of initial parameters (if the user input procedure option is selected)
Option to enable automatic readjustment of Ka and Kp for friction angle and design approach changes (if the option B is selected)
Option to enable automatic readjustment of Ka and Kp for surface slope angle modifications (if the option B is selected)
Option to enable automatic readjustment of Ka and Kp for earthquake effects (if the option B is selected)
Option to adjust Ko when friction angle changes
Option to allow modification of Ka and Kp on each stage and each design section.

- **Include wall friction:** In this area we can choose to include wall friction to the calculations on any wall side (resisting and driving side), as well as define how the wall friction is to be calculated.

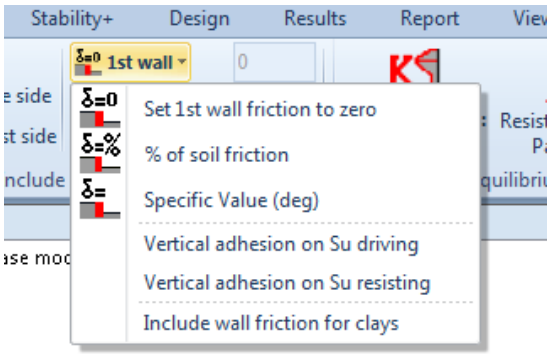


Figure 2.9.9: Include wall friction.

Option to set the wall friction to zero
Option to add wall friction as a percentage of soil friction (0 to 100%)
Option to use a specific value for the wall friction
Choice to set vertical adhesion on Su driving (limit equilibrium analysis)
Choice to set vertical adhesion on Su resisting (limit equilibrium analysis)
Choice to also include wall friction for clays

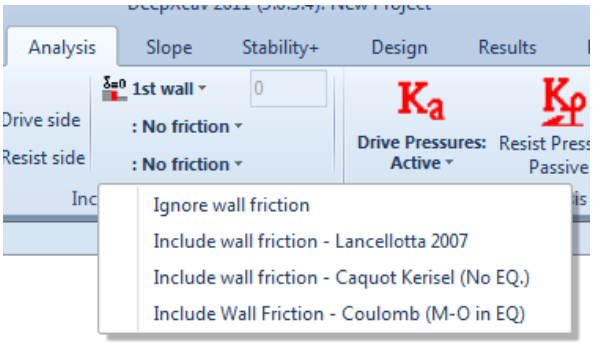


Figure 2.9.10: Include wall friction for driving and resisting side.

Option ignore wall friction
Option to use Lancellotta 2007 method for wall friction
Option to use Caquot Kerisel method for wall friction
Option to use Coulomb method for wall friction

- **Conventional analysis methods – Drive Pressures:** In this area we can define the type of lateral earth pressures used on the driving side for a limit equilibrium analysis.

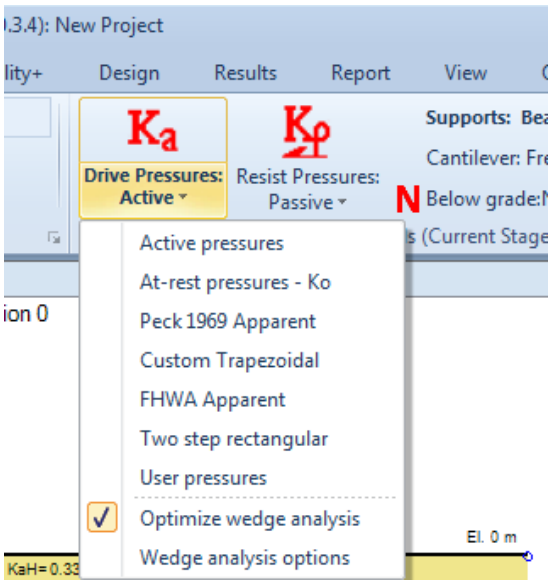


Figure 2.9.11: Active pressures.

➔ Active pressures: By choosing the active pressures option we can choose to use active pressure for the driving side.

➔ At-rest pressures: By choosing the at-rest pressures option we can choose to use at-rest pressures for the driving side.

➔ Peck 1969 apparent: By choosing the Peck pressures option and by pressing the button

, the Peck pressure options dialog appears.

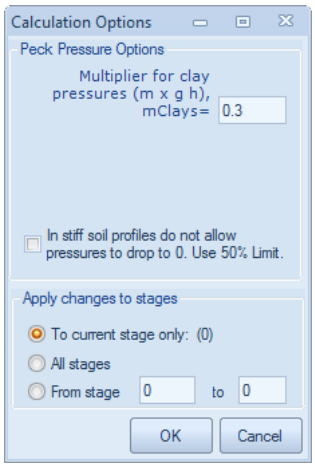


Figure 2.9.12: The peck pressures options dialog.

In the Peck pressures options dialog we can define the multiplier for clay pressures mClays, and in addition there is a choice in stiff soil profiles not to allow pressures to drop to zero.

➔ Custom trapezoidal: By choosing the Custom trapezoidal pressures option and by pressing the button , the Trapezoidal pressure options dialog appears.

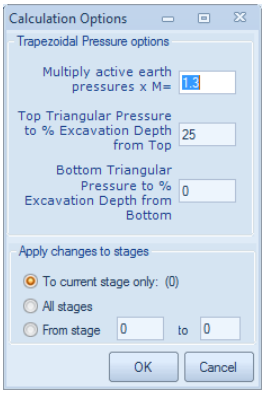


Figure 2.9.14: The trapezoidal pressures options dialog.

In the trapezoidal pressures options dialog we can define the multiplier for the active earth pressures M, the top triangular pressure to a percentage of excavation depth from top and the bottom triangular pressure to a percentage of excavation depth from bottom.

➔ FHWA apparent: By choosing the FHWA pressures option and by pressing the button , the FHWA pressure options dialog appears.

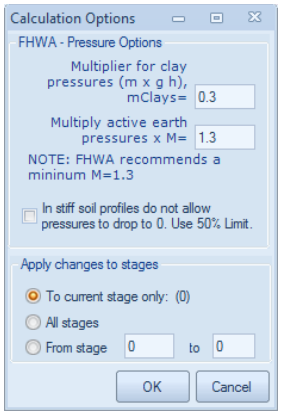


Figure 2.9.15: The FHWA pressures options dialog.

Here we can define the multiplier for clay pressures mClays, the multiplier for the active earth pressures M and in addition there is a choice in stiff soil profiles not to allow pressures to drop to zero.

➔ Two step rectangular: By choosing the Two step rectangular pressures option and by pressing the button , the Two step rectangular pressure options dialog appears. With this option lateral earth pressures above the excavation are calculated as M1 x Hexc above the water table and as Mz x Hexc below the water table.

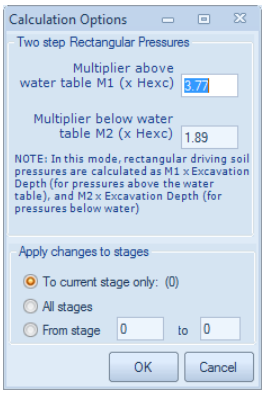

Calculation Options
Two step Rectangular Pressures
Multiplier above water table M1 (x Hexc)
3.77
Multiplier below water table M2 (x Hexc)
1.89
NOTE: In this mode, rectangular driving soil pressures are calculated as M1 x Excavation Depth (for pressures above the water table), and M2 x Excavation Depth (for pressures below water)
Apply changes to stages
To current stage only: (0)
All stages
From stage
0
to
0
OK
Cancel

Figure 2.9.16: The Two step rectangular pressures options dialog.

Here we can define the multiplier for above water table M1 and the multiplier below water table M2.

➔ User pressures: By choosing the user pressures option and by pressing the button the custom pressures dialog appears.

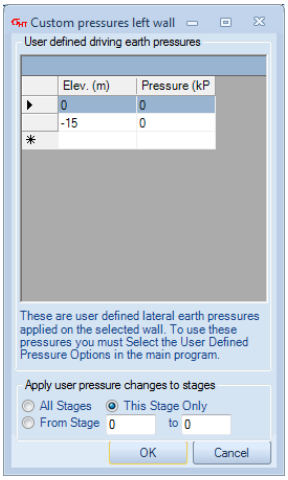


Figure 2.9.17: The custom pressures dialog.

Here we can define the values of pressures at several elevations.

➔ We can choose whether to optimize the wedge analysis or not.

➔ Wedge analysis options: By choosing the wedge analysis options, the wedge analysis options dialog appears.

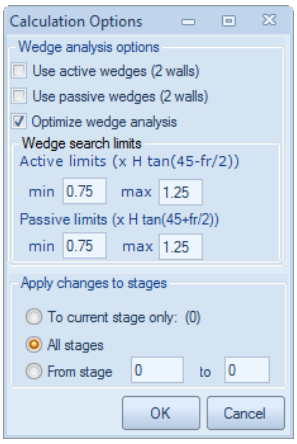


Figure 2.9.18: The wedge analysis options dialog.

The following table presents the options that are available in the wedge analysis options dialog:

Choice to use active wedges for the two walls
Choice to use passive wedges for the two walls
Choice to optimize wedge analysis
Define the minimum and maximum active and passive wedge search limits. These limits are used when the Optimize wedge analysis option is selected.

- **Conventional analysis methods – Resist Pressures:** In this area we can choose the type of lateral earth pressures used on the resisting side.

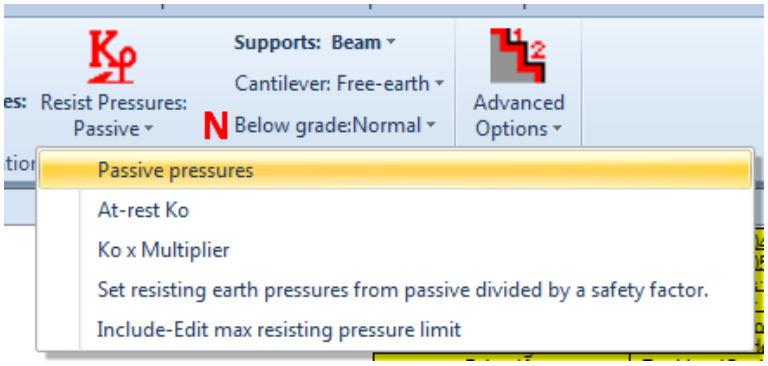


Figure 2.9.19: Resist pressures.

Choice to use passive pressures at the resisting side (Figure 2.9.20)
Choice to use at-rest pressures at the resisting side
Choice to multiply the passive pressures with a multiplier (Figure 2.9.21)
Choice to set resisting earth pressures from passive divided by a safety factor. (Figure 2.9.22)
Choice to include-edit a maximum passive pressure limit (Figure 2.9.23)

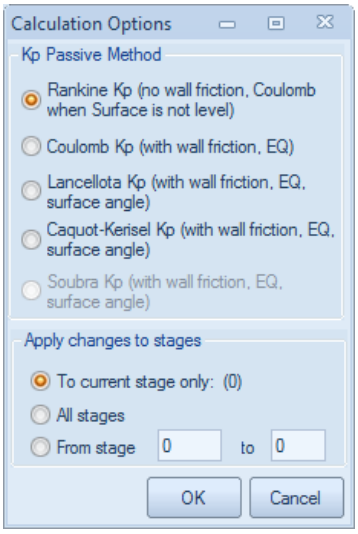

Calculation Options
Kp Passive Method
Rankine Kp (no wall friction, Coulomb when Surface is not level)
Coulomb Kp (with wall friction, EQ)
Lancellota Kp (with wall friction, EQ, surface angle)
Caquot-Kerisel Kp (with wall friction, EQ, surface angle)
Soubra Kp (with wall friction, EQ, surface angle)
Apply changes to stages
To current stage only: (0)
All stages
From stage 0 to 0
OK
Cancel

Figure 2.9.20: Passive pressures dialog.

The following table presents the options that are available in the passive pressures dialog:

Choice to use Rankine Kp (no wall friction)
Choice to use Coulomb Kp
Choice to use Lancellotta Kp
Choice to use Caquot-Kerisel Kp
Choice to use Soubra Kp

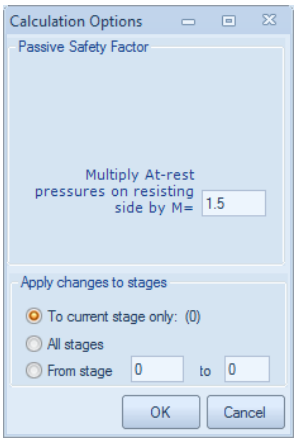


Figure 2.9.21: Passive pressures Multiplier dialog (for at-rest pressures).

Here we can define the multiplier M for the resisting at rest pressures.

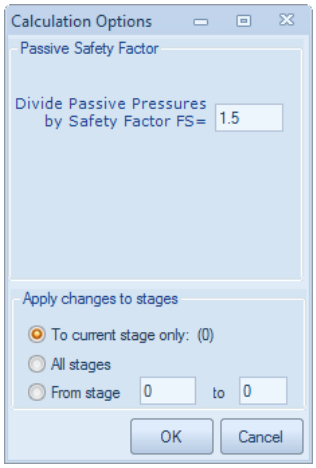


Figure 2.9.22: Passive pressures Safety factor dialog.

Here we can define the safety factor FS. Passive earth pressures will be divided by this FS to obtain the resisting earth pressures.

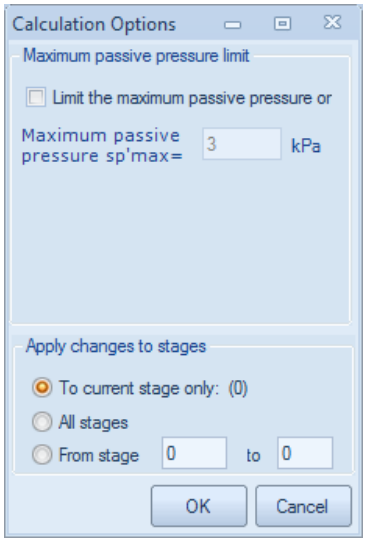


Figure 2.9.23: Maximum passive pressure limit dialog.

Here we can define the maximum passive limit sp'max.

- **Conventional analysis methods – Cantilever:** In this area we can choose the cantilever method (Free or Fixed earth)

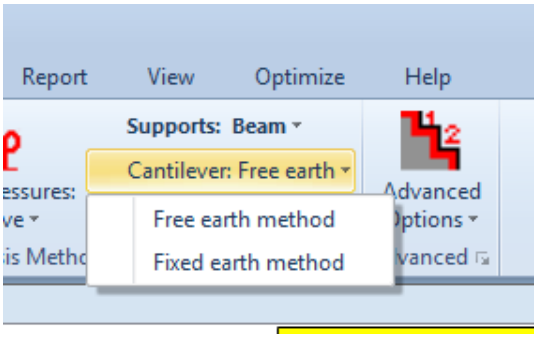
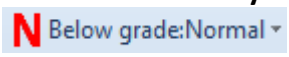


Figure 2.9.24: Cantilever method.

- **Conventional analysis methods – below grade pressures:** By clicking on the button 

, we user can choose the below grade pressure method. In general it is recommended to keep the Normal option selected. The = 0 pressures option will apply zero earth pressures below the excavation, while the Ka-Active option forces active earth pressures below excavation.

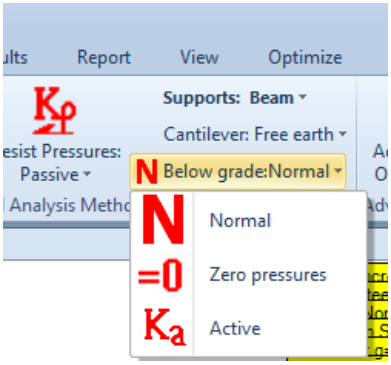
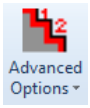


Figure 2.9.25: Below grade pressures..

- **Advanced options:** By pressing the button 

, we can find some advanced choices regarding the active and passive coefficients.

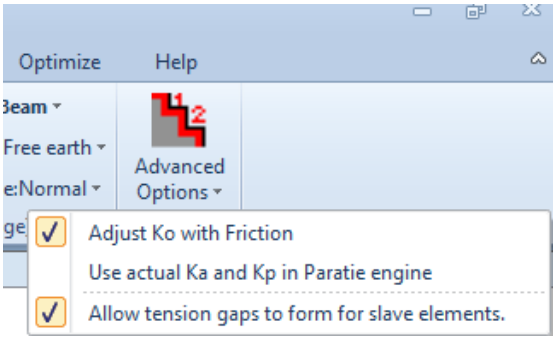


Figure 2.9.26: Advanced options.

Adjust Ko with friction	Adjust Ko when ϕ' changes due to a material change (Eurocode 7)
Use actual Ka and Kp in paratie engine	Uses the theoretically consistent equations for Ka and Kp
Allow tension gaps to form for slave elements	Allows the formation of tension gaps between master and slave nodes when slave elements have been defined (supports or wall elements). This option can be used when liner walls have to be modeled

2.10 Slope Stability menu

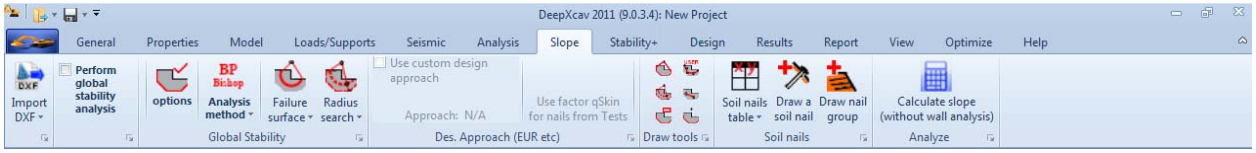


Figure 2.10.1: The Slope tab menu.

- **Import from DXF:** By pressing the button

, we can import a DXF file in DeepXcav and use the defined geometry. Some limitations in the import apply.

- **Options:** By pressing the button

, the slope stability options dialog appears. Here we can define the method and options for the slope stability analysis. **The properties contained in this dialog are described in paragraph 3.13.**

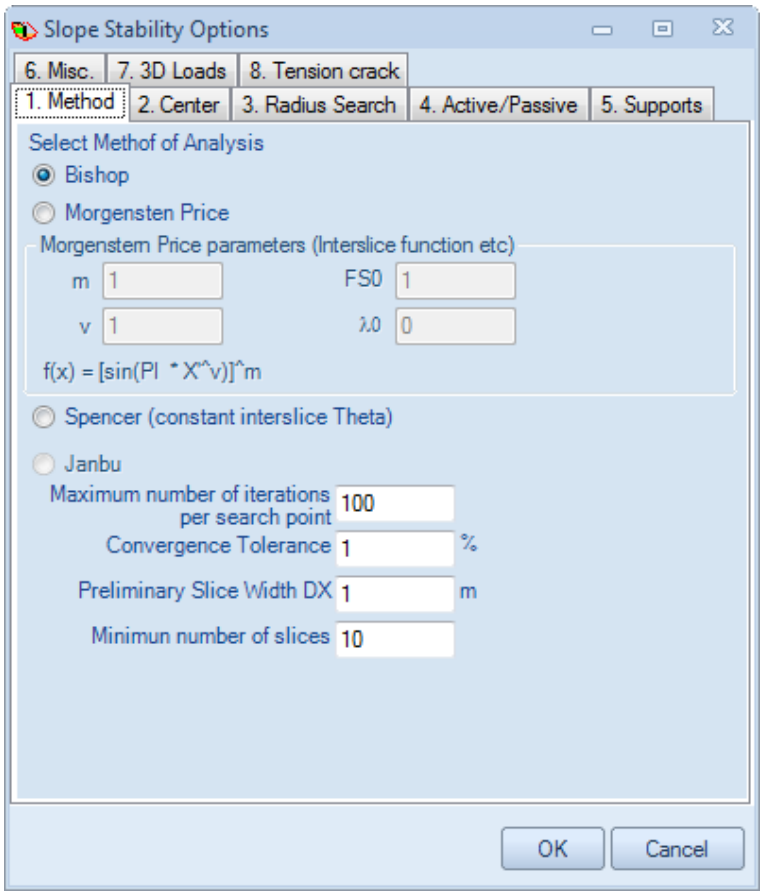


Figure 2.10.2: The Slope stability options dialog.

- **Analysis method:** By pressing the button

, we can define the slope stability analysis method.

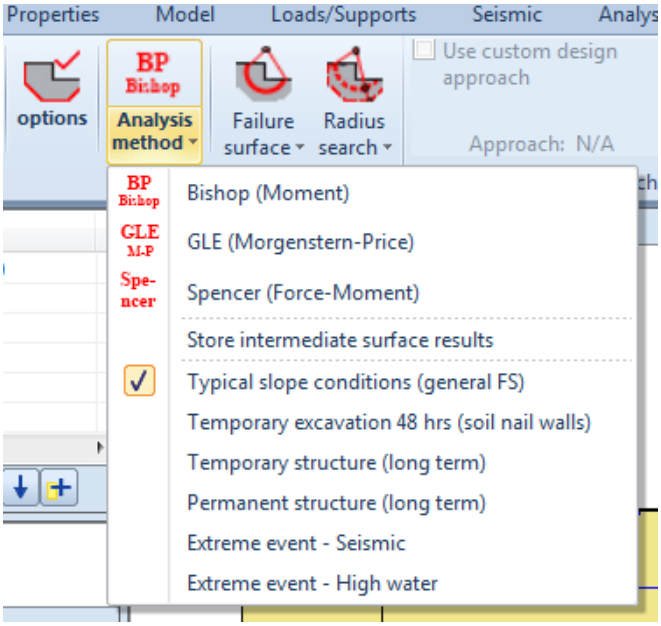


Figure 2.10.3: Slope stability analysis methods.

Bishop	Use the Bishop simplified method for analyzing slope stability (only circular surfaces)
GLE	Use the Morgenstern – Price method for analyzing slope stability
Spencer	Use the Spencer method for analyzing slope stability
Store intermediate surface results	With this option the program will store all intermediate slope stability surfaces and safety factors

In some design codes (such as AASHTO) minimum safety factors are defined for different conditions (long term, short term etc). These minimum safety factors will appear once the analysis has been performed.

Typical slope conditions	Uses a general safety factor
Temporary excavation	Used for an excavation designed to last 48 hours
Temporary structure	Used for a temporary long term structure
Permanent structure	Used for a permanent long term structure
Extreme event - seismic	Describes an extreme event (seismic)
Extreme event – high water	Describes an extreme event (high water)

- **Failure surface:** By clicking on the button

, we can define the shape of the failure surface.

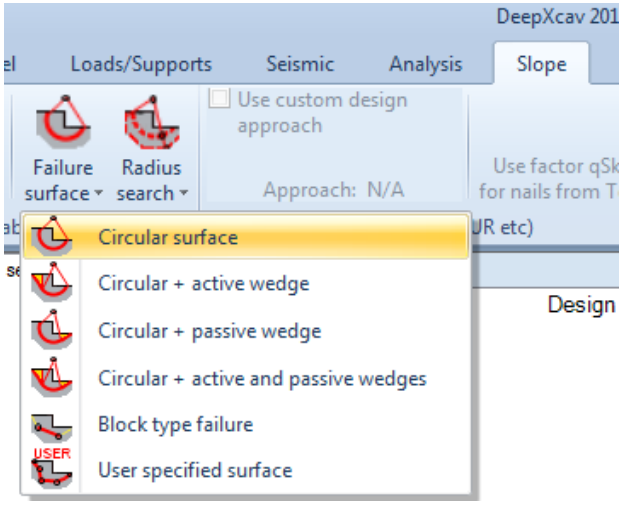
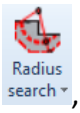


Figure 2.10.4: Sliding surface shape options.

- **Radius search:** By clicking on the button 

, we can define some options for the radius search.

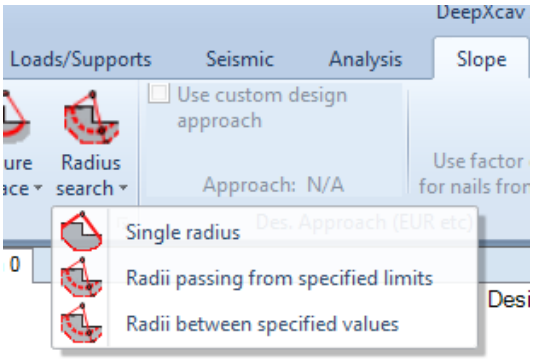


Figure 2.10.5: Radius search options.

Single radius	One radius
Radii passing from specified limits	Radii defined from two points
Radii between specified values	Radii defined from a first radius value to a second radius value

Draw tools: The user can use this toolbar to

- draw the sliding surface or draw the search rectangle etc. The icons are explained in the table below:

Icon	Description
	Draw a single radius failing surface.
	Draw a custom failure surface by clicking on a series of points on the screen.
	Define the start and end points where the failing surface is to pass from.
	Define the base limits for s block wedge analysis.
	Draw a rectangular box that contains the centers of the failure surfaces
	Define a single point for the center of the sliding surface (circular type surfaces).

- **Soil nails:** In this area we can define the properties of soil nails used within the model.

  - **Soil nails table:** By pressing the button

, the Soil nails table appears. This table contains the basic dimensions and properties of each soil nail. By clicking the button, we can choose some further options regarding the slope stability analysis with soil nails. This options can also be set for each soil nail separately.

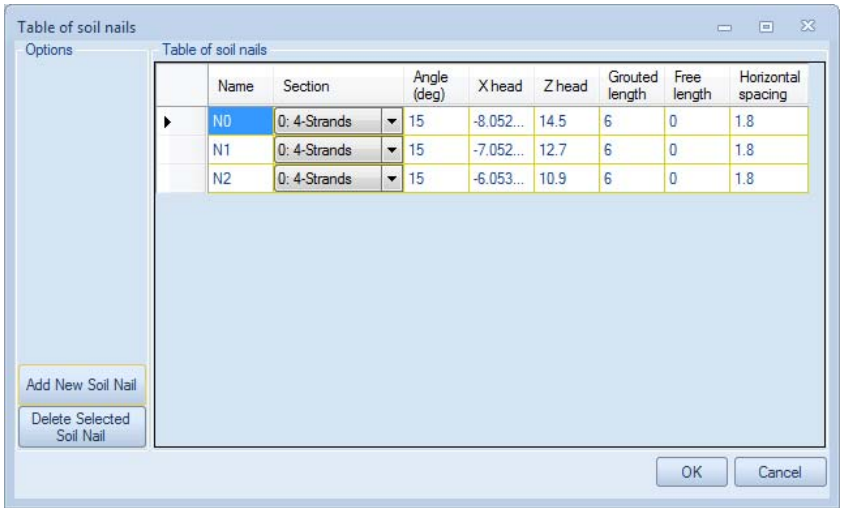


Figure 2.10.6: Soil nails table dialog.

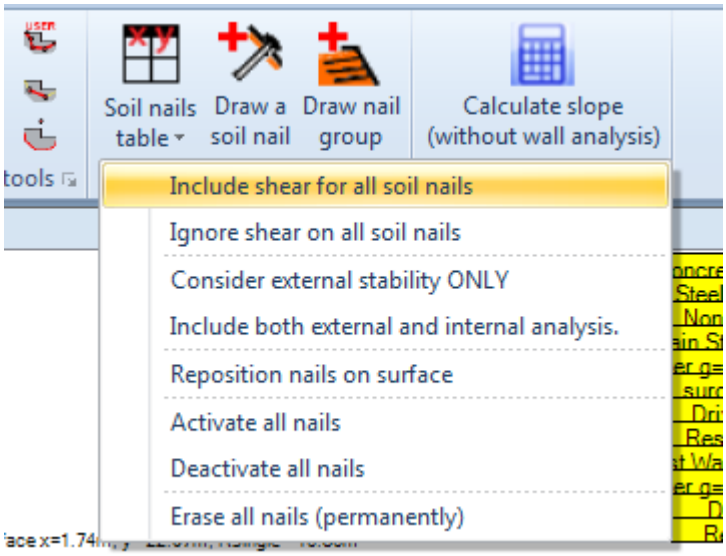

Soil nails table
Draw a soil nail
Draw nail group
Calculate slope (without wall analysis)
tools
Include shear for all soil nails
Ignore shear on all soil nails
Consider external stability ONLY
Include both external and internal analysis.
Reposition nails on surface
Activate all nails
Deactivate all nails
Erase all nails (permanently)

Figure 2.10.7: Soil nails options.

The following table contains the soil nail options that are available:

Include shear for all soil nails
Ignore shear for al soil nails
Consider only external stability
Include both external and internal analysis
Reposition nails on surface
Activate all nails
Deactivate all nails
Erase all nails permanently

External stability assumes that the soil nail fails by pullout of the nail beyond the failure surface. When internal analysis is included, the possibility on a nail failing from the head face (internal side) is also considered.

- **Draw a soil nail:** By pressing the button

, we can add a single soil nail to the model. To add a nail first click on the surface (head) and then on the end point of the nail.

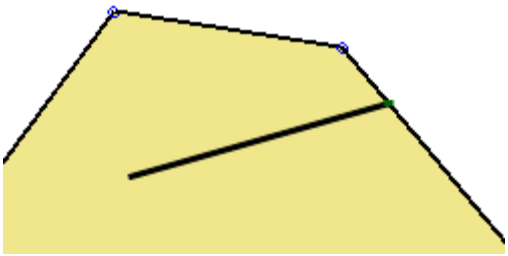


Figure 2.10.7: Add a soil nail.

- **Draw a nail group:** By pressing the button

, we can add a group of soil nails to the model. To do so, click on the first point (close to the surface) and then click on the second point.

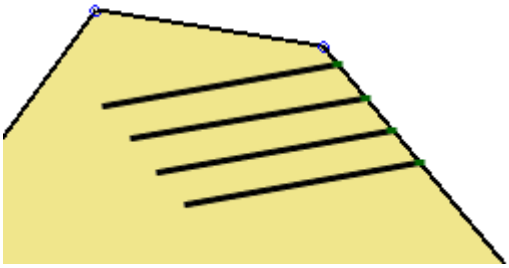
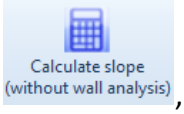
- 

Figure 2.10.8: Add a soil nail group.

- **Analyze:** By pressing the button 

, the program will perform a slope stability analysis and calculate the factor of safety of the slope, without performing a wall analysis. To use this option the general analysis must be performed and general settings not changed.

2.11 Stability+ menu

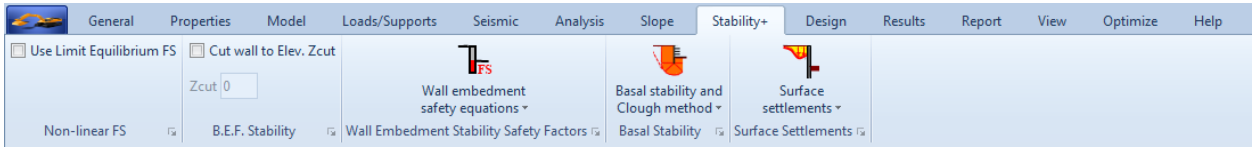


Figure 2.11.1: The Stability+ tab menu.

In this area we can control options regarding the calculation of wall embedment safety factors, basal stability and surface settlement.

- **Limit equilibrium FS:** In this area we can select to use a conventional factor of safety when the non-linear analysis is performed (Passive pressures will be calculated from LEM analysis).

- **Non-linear FS:** In this area we can force the passive and active force limits to be calculated from LEM analysis.

- **B.E.F. Stability:** In this area we can use a Zcut command in the non-linear analysis. This command deactivates all wall nodes below this elevation.

- **Wall embedment stability equations:** In this area we can control which Factors of Safety regarding the wall embedment are included within the limit equilibrium analysis (Figure 2.11.2.)

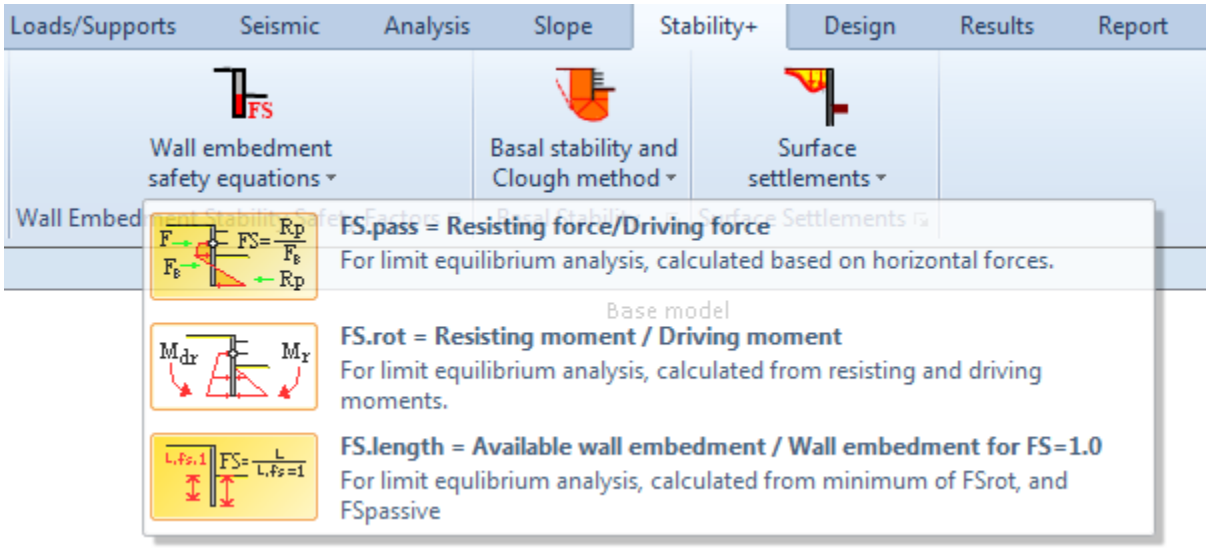

Loads/Supports
Seismic
Analysis
Slope
Stability+
Design
Results
Report
Wall embedment safety equations
Basal stability and Clough method
Surface settlements
FS.pass = Resisting force/Driving force
For limit equilibrium analysis, calculated based on horizontal forces.
Base model
FS.rot = Resisting moment / Driving moment
For limit equilibrium analysis, calculated from resisting and driving moments.
FS.length = Available wall embedment / Wall embedment for FS=1.0
For limit equlibrium analysis, calculated from minimum of FSrot, and FSpassive

Figure 2.11.2: The wall embedment safety equations.

- **Basal stability and Clough method:** In this area we can define whether to estimate maximum wall displacements according to the semiempirical Clough method. We can select from options presented in Figure 2.11.3. By clicking on the button , the dialog in Figure 2.11.4 appears.

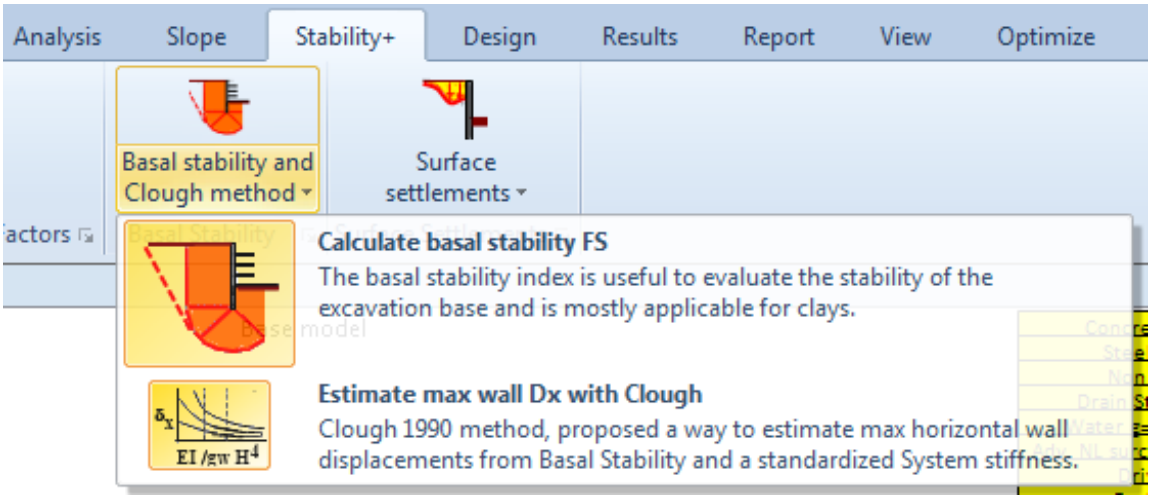

Analysis
Slope
Stability+
Design
Results
Report
View
Optimize
Basal stability and Clough method
Surface settlements
Calculate basal stability FS
The basal stability index is useful to evaluate the stability of the excavation base and is mostly applicable for clays.
Estimate max wall Dx with Clough
Clough 1990 method, proposed a way to estimate max horizontal wall displacements from Basal Stability and a standardized System stiffness.

Figure 2.11.3: Basal stability and Clough options.

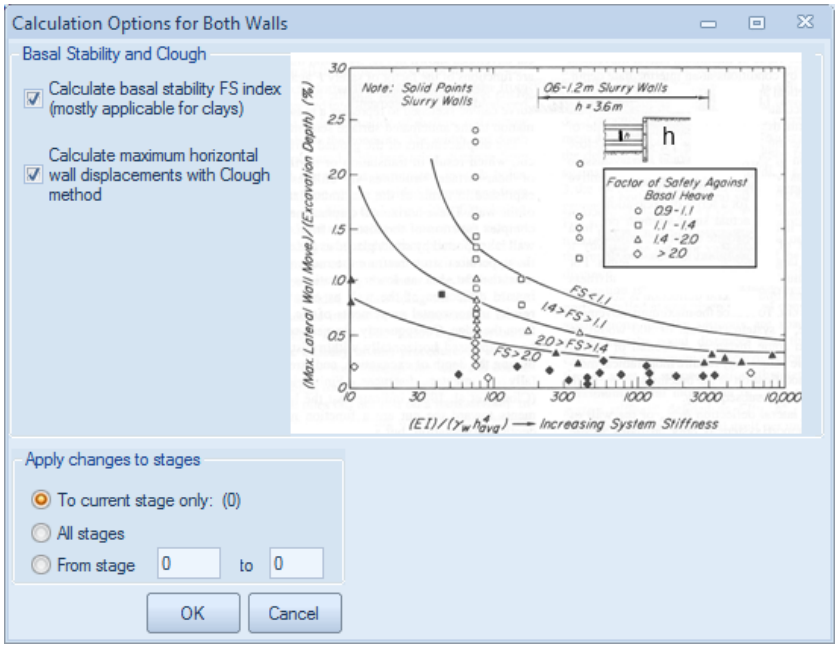


Figure 2.11.4: Basal stability and Clough options dialog.

In this form we can choose to calculate the basal stability FS, and to calculate the maximum horizontal wall displacements with Clough method.

- **Surface settlements:** In this area we can define whether to estimate surface settlements or not. We can select the options that are presented in Figure 2.11.5. By clicking on the button , the dialog in Figure 2.11.6 appears. For LEM analysis, the original method uses the Clough calculated displacements with some modification factors. These modification factors can be enabled by selecting the "Use modifications for Clough" option.

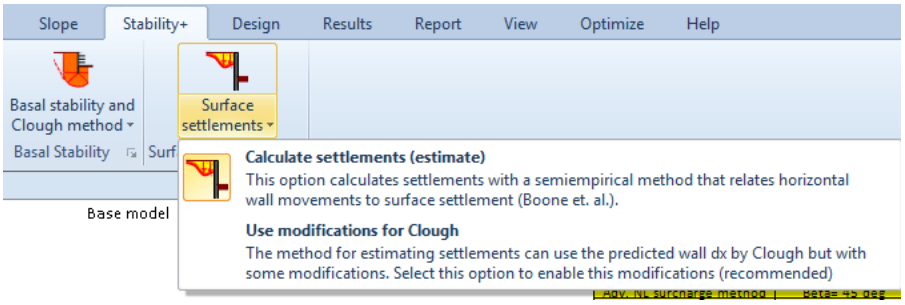

Slope
Stability+
Design
Results
Report
View
Optimize
Help
Basal stability and Clough method
Basal Stability
Surface settlements
Base model
Calculate settlements (estimate)
This option calculates settlements with a semiempirical method that relates horizontal wall movements to surface settlement (Boone et. al.).
Use modifications for Clough
The method for estimating settlements can use the predicted wall dx by Clough but with some modifications. Select this option to enable this modifications (recommended)

Figure 2.11.5: The surface settlement options.

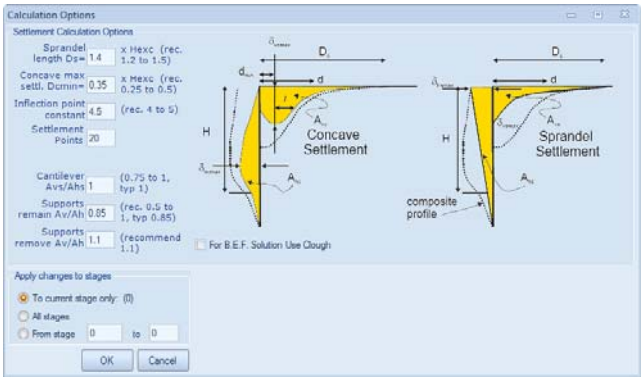


Figure 2.11.5: The settlement calculation options dialog.

The following table presents the parameters that we can define in this dialog:

Define the Sprandel length Ds in terms of the excavation height
Define the concave maximum settlement Dcmin
Inflection point constant
Settlement points (where settlement is calculated)
Cantilever Avs/Ahs (vertical Sprandel volume to horizontal displaced Sprandel volume)
Supports remain Av/Ah (ratio of vertical to horizontal volume)
Supports remove Av/Ah (ratio of vertical to horizontal volume when the supports are removed)
Option to use Clough method in BEF analysis. This option forces the program to calculate the horizontal displacement volume with the Clough method.

For more information please see theory manual.

2.12 Design menu

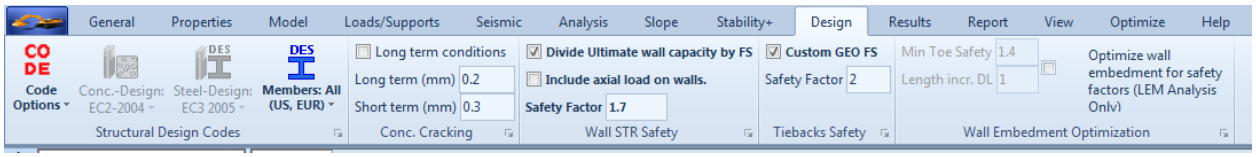


Figure 2.12.1: The Design tab menu.

- **Code options:** By clicking on the button

, we can define which structural code's settings to apply in analysis. These code settings control structural codes and other options.

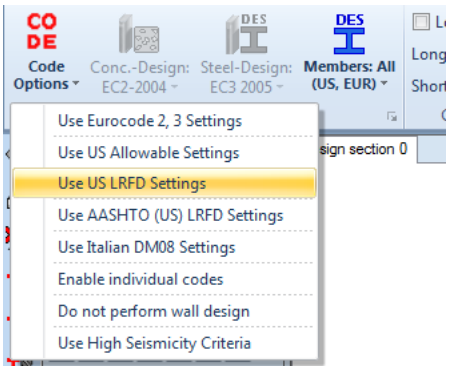


Figure 2.12.2: Code options.

The following options are available:

Use Eurocode 2, 3 settings	Apply Eurocode settings to the design
Use US allowable settings	Apply US allowable settings to the design
Use US LRFD settings	Apply US LRFD settings to the design
Use AASHTO (US) LRFD settings	Apply AASHTO LRFD settings to the design
Enable individual codes	Enables the possibility to use different codes for concrete and steel design
Do not perform wall design	This cancels the wall design
Use high seismic criteria	Uses high seismic criteria for structural design (when applicable)

- **Members:** By clicking on the button

, we can define which standards to be used in selecting steel members and reinforcement bars.

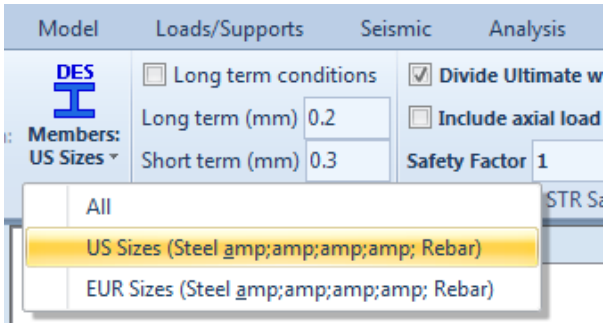


Figure 2.12.3: Steel member code options.

- **Concrete cracking:** In this area we can define the allowable cracking widths when serviceability design is performed. The long term limit will be used when "Long term conditions" option is selected.

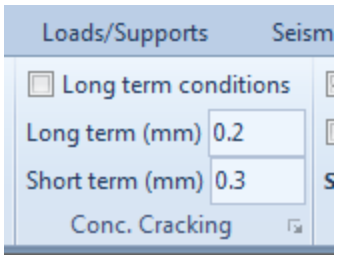


Figure 2.12.4: Concrete cracking options.

- **Wall STR Safety:** In this area we can define the wall structural Factor of Safety. This factor is used to divide the yield wall capacity and produce the design wall capacity.

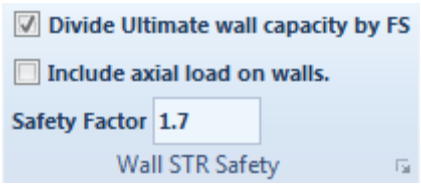


Figure 2.12.5: Wall STR safety options.

- **Include axial load on walls:** With this option we can include axial loads on walls from tiebacks etc. Please note that these loads are not factored.

- **Tiebacks Safety:** In this area we can define the geotechnical Factor of Safety used for calculating the design pullout capacity of a tieback. Please note that certain design codes will overwrite this setting (EC7, DM08 etc).

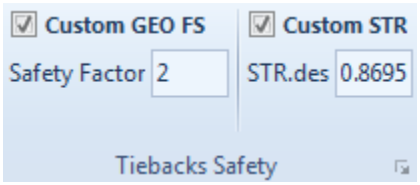


Figure 2.12.6: Tieback safety options.

- **Wall embedment optimization:** In this area we can choose to optimize wall embedment safety factors regarding the conventional analysis. Please note that this option should only be used when one wall is present.

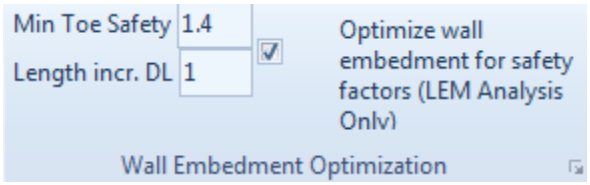

Min Toe Safety 1.4
Length incr. DL 1
Optimize wall embedment for safety factors (LEM Analysis Only)
Wall Embedment Optimization

Figure 2.12.7: Wall embedment optimization options.

2.13 Results menu

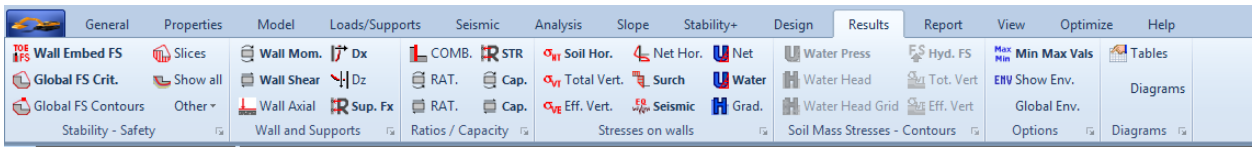
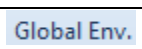
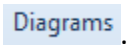


Figure 2.13.1: The Results tab menu.

This tab contains a list of results that can be either viewed on the model or presented in diagrams or tables. Results can be presented when the analysis has been completed.

Icon	Description
	Wall embedment Safety Factors
	Global Slope Stability Factor of Safety
	Global Slope Factor of Safety Contours
	Results for slope stability slices
	Show all slope stability results (when intermediate surfaces are stored)
	Wall bending moment
	Wall shear
	Wall Axial force
	Wall Displacement (horizontal)
	Surface settlements
	Support reactions
	Combined capacity ratio (Moment and Axial force combined check)
	Moment capacity ratio = Actual moment/Design wall moment capacity ≤ 1.0 Good
	Shear capacity ratio = Actual shear/Design wall shear capacity ≤ 1.0 Good
	Structural capacity ratios for supports = Actual load/design capacity
	Wall moment capacity
	Wall shear capacity
σ_{HT}	Effective horizontal soil pressure
σ_{VT}	Total vertical stress

	Effective vertical stress
	Net total wall pressures
	Wall surcharge
	Seismic pressures
	Net water pressure
	Water pressure on walls
	Hydraulic gradient
	Minimum – Maximum values
	Local result envelopes: shows envelopes of results for current design section
	Global envelope: shows envelopes of results for all linked design sections
	Result tables: Launches a results table
	Water pressure shadings (when a flow net analysis has been performed)
	Water head shadings (when a flow net analysis has been performed)
	Water head grid (when a flow net analysis has been performed)
	Hydraulic safety factor (when a flow net analysis has been performed)
	Total vertical stress shadings (when a flow net analysis has been performed)
	Effective vertical stress shadings (when a flow net analysis has been performed)

In addition, we can choose to see diagrams of the results by pressing the button  .

2.14 Report menu

From the Report tab we can control options for viewing reports in pdf or word formats as well as see summary tables of all calculations.

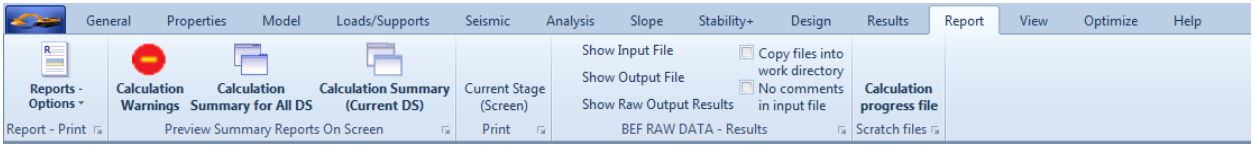


Figure 2.14.1: The Reports tab menu.

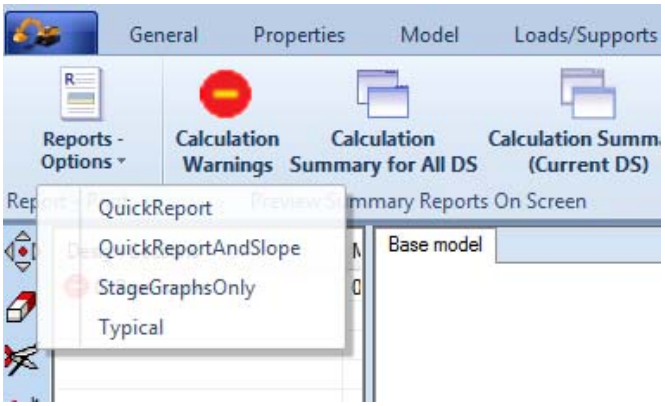


Figure 2.14.2: Report options.

Option to create a quick report
Option to create a quick report and include slope stability
Option to create a only stage graphs report
Option to create a typical report

- **Report - print:** By pressing the button

, the Reports dialog appears. In this dialog we can choose what parts of calculations and results we wants to be included in the reports and the type of file to be exported.

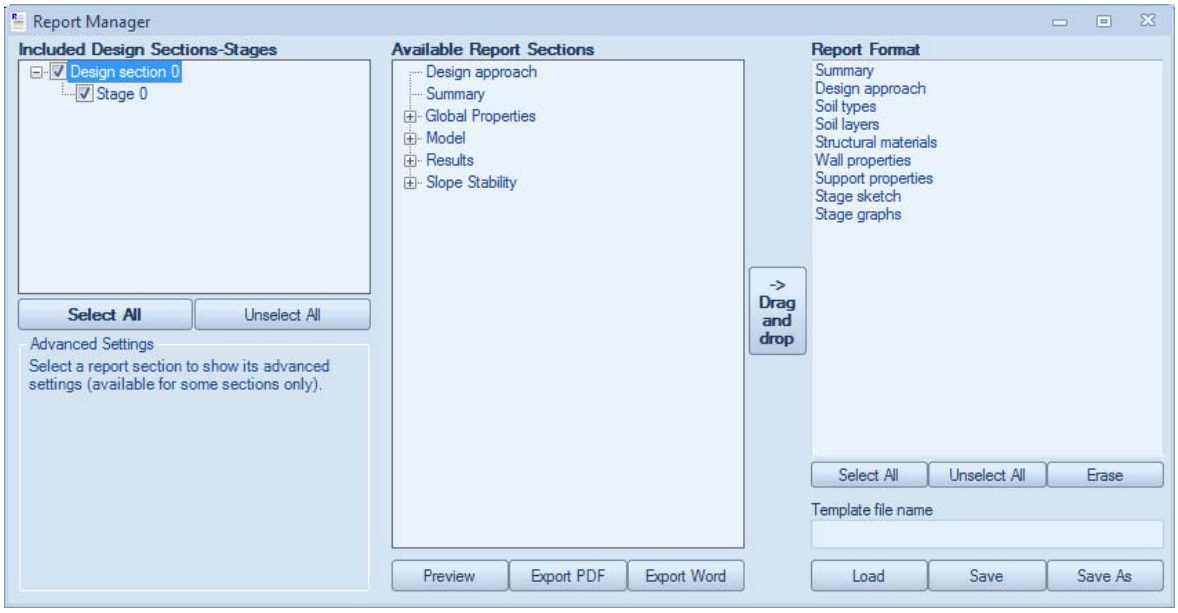


Figure 2.14.3: The Report dialog.

- **Calculation Warnings:** By pressing the button

, the Calculation warnings dialog appears. Here we can see gemeral warnings that are generated when calculations have been performed.

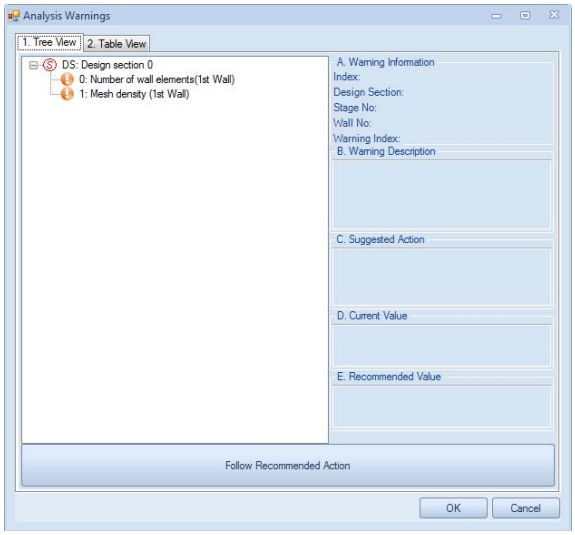
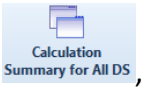
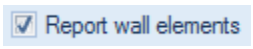


Figure 2.14.4: Calculation warnings.

- **Calculation summary for all design sections – current design section:** By pressing the button 

, the Calculation Summary dialog appears. This dialog presents a list of results for all and each one individual design section. Select the

option to show available results for one design section. The 

option controls if results for additional wall elements are included in the model.

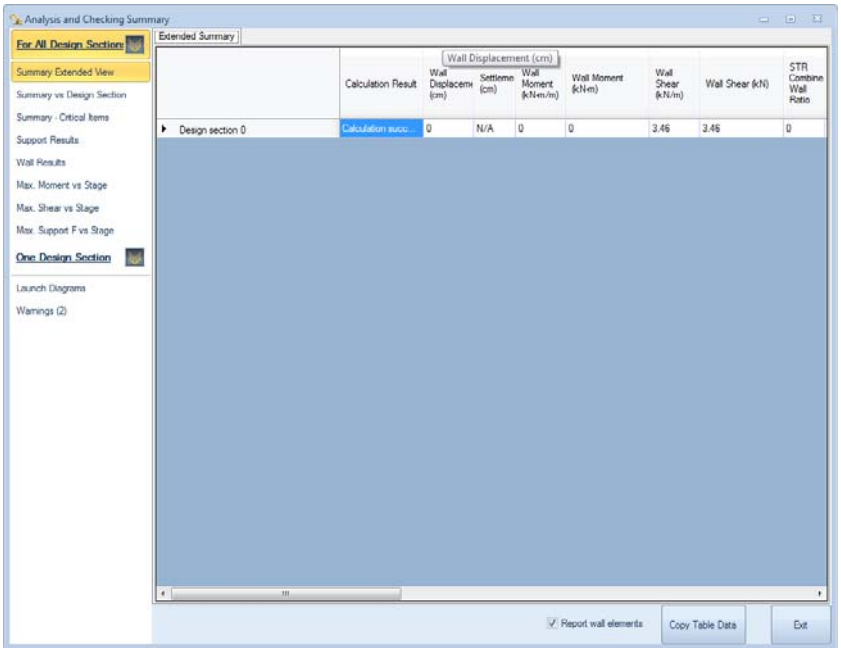


Figure 2.14.5: Calculation summary dialog.

- **Current stage (screen):** This creates a preview of the model as it appears on the screen. (This preview can then be printed).

- **BEF Raw data - results:** By pressing one of the buttons here, a .TXT document is created, containing the input or output data of the model for the non-linear analysis.

- **Calculation progress file:** By pressing one of the buttons here, a .TXT document is created, containing the solution progress file that contains basic progress calculations during the analysis.

2.15 View menu

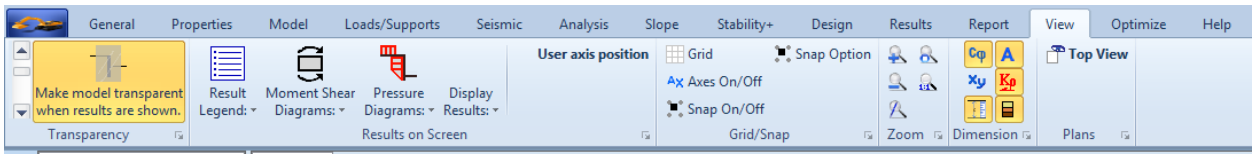
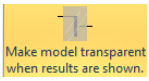


Figure 2.15.1: The View tab menu.

- **Transparency:** By pressing the button 

we can make the model transparent when results are shown. The transparency can be adjusted from the vertical bar.

- **Result legend:** By pressing the button

we can turn the result legend on or off.

- **Moment shear diagrams:** By pressing the button

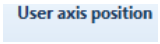
we can change the view settings of the moment and shear diagrams.

- **Pressure diagrams:** By pressing the button

we can change the view settings of the pressure diagrams.

- **Display results:** By pressing the button

we can change the display settings of the results.

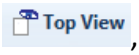
- **User axis position:** By pressing the button 

we can change the axis position (elevation).

- **Grid/snap:** By pressing the button

we can change the view options of the snap and axis or show a grid.

- **Zoom and Dimension:** These icons are explained in chapter 2.3.1.

- **Top view:** By pressing the button 

, a top plan view of the site is created. This allows the user to see the position of the design section and the location of 3D objects (i.e. buildings, footing loads etc).

2.16 Optimize menu

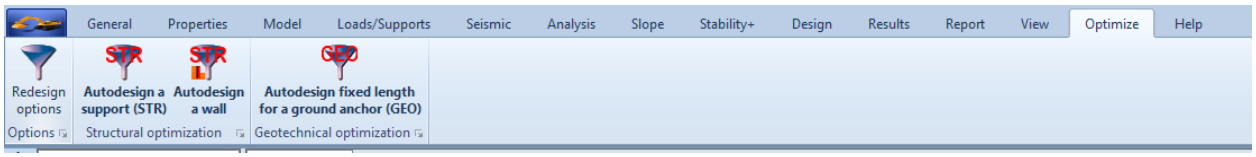
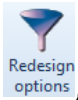


Figure 2.16.1: The Optimize tab menu.

- **Redesign options:** By pressing the button 

, the structural redesign – optimization dialog appears. Here we can define the redesign options for walls and supports.

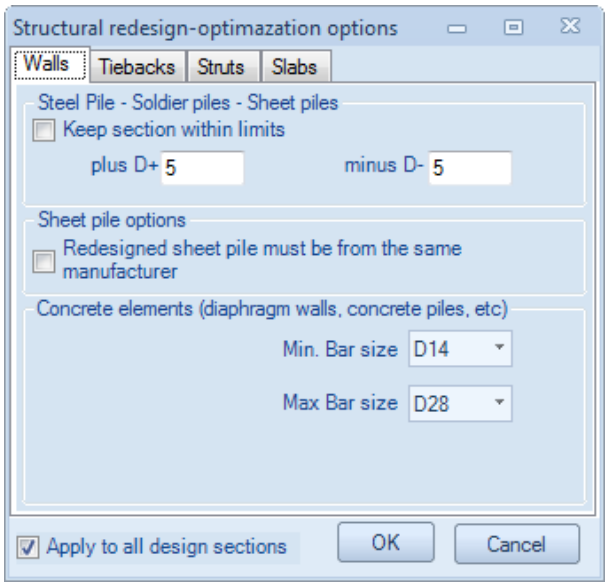


Figure 2.16.2: The redesign - optimization dialog – Walls tab.

The table below presents the options that are available in the walls tab:

Option to keep section within limits: Keeps the optimized section within the specified limits from the original section.
Define the maximum (D+) and minimum (D-) limit, if the previous option is selected
Choice that the redesigned sheet pile wall must be of the same manufacturer
Minimum and maximum bar sizes for diaphragm walls and concrete elements

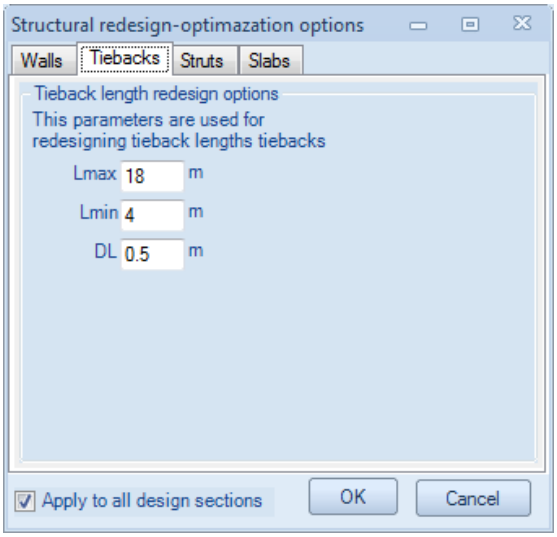


Figure 2.16.3: The redesign - optimization dialog – Tiebacks tab.

The table below presents the options that are available in the tiebacks tab:

Define the maximum length Lmax
Define the minimum length Lmin
DL = Length increment for optimization

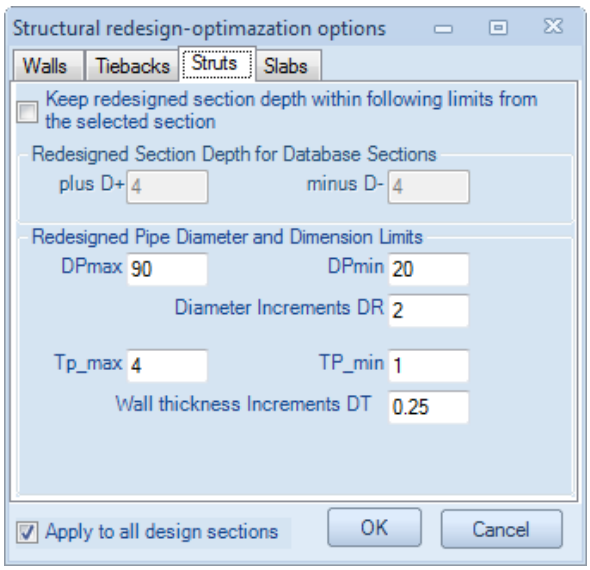


Figure 2.16.4: The redesign - optimization dialog – Struts tab.

The table below presents the options that are available in the struts tab:

Option to keep section within limits (from original dimensions)
Define the maximum (D+) and minimum (D-) limit, if the previous option is selected
Define the maximum (DPmax) and minimum (DPmin) pipe diameter
Define the maximum (Tp_max) and minimum (Tp_min) pipe diameter

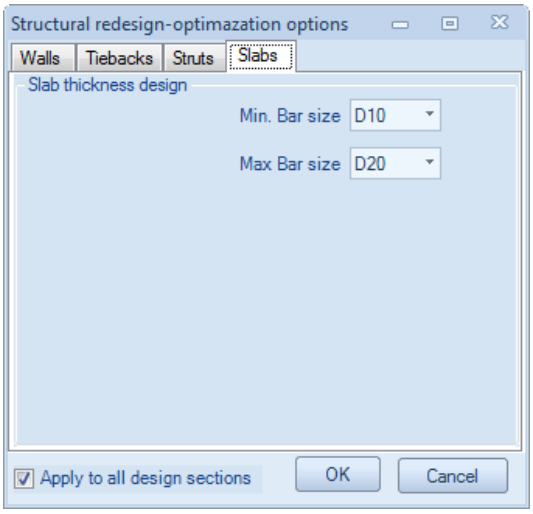


Figure 2.16.5: The redesign - optimization dialog – Slabs tab.

The table below presents the options that are available in the slabs tab:

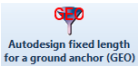
Minimum and maximum bar sizes for slabs

- **Autodesign a support:** By pressing the button

after the calculations are performed and choosing a support, the program performs an automatic structural optimization of the selected support.

- **Autodesign a wall:** By pressing the button

after the calculations are performed and choosing a wall, the program performs an automatic structural optimization of the selected wall.

- **Autodesign fixed length for a ground anchor:** By pressing the button 

after the calculations are performed and choosing a ground anchor, the program performs an automatic optimization of the fixed length of the selected ground anchor.

2.17 Help menu

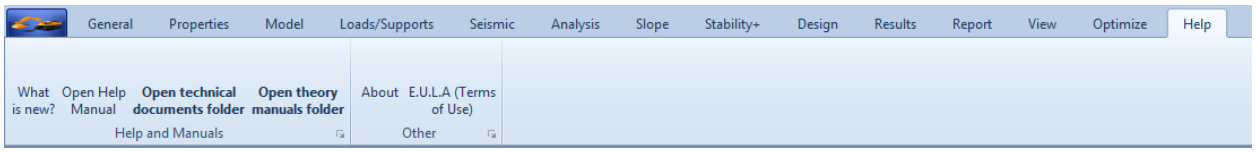


Figure 2.17.1: The Help tab menu.

- **What is new:** By pressing the button What is new?, we can check what is new in the latest version of DeepXcav.

- **Open help manual:** By pressing the button Open Help Manual, we can open the DeepXcav help file.

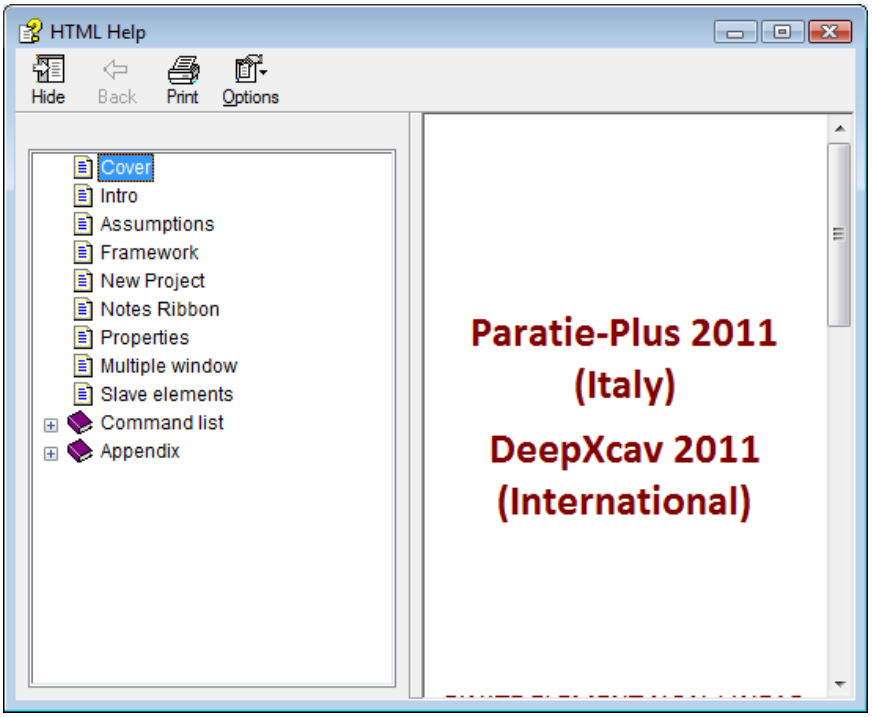


Figure 2.17.2: The DeepXcav help file.

- **Manuals:** By pressing the buttons Open technical documents folder and Open theory manuals folder, the folders that contain the technical documentation and the theory manual of DeepXcav are displayed.

- **Other:** By pressing the button About E.U.L.A (Terms of Use), the user can read the terms of use of DeepXcav.

CHAPTER 3: DATA ENTRY

\

3.1 Data entry: General

DeepXcav is a very user-friendly software program and includes powerful features and versatile options. The main interface is shown below. The general philosophy in creating an excavation model in DeepXcav is:

1) Specify the global coordinates

2) Specify the soil types and properties

3) Specify the layers

4) Create a generalized water table

5) Specify the retaining wall system (soldier pile, sheet pile, secant pile, tangent pile, and diaphragm wall options).

6) Create a database of support members (tiebacks, struts, or concrete slabs)

7) Add a stage and draw in new supports

8) Modify the stage elevations

9) Analyze the project.

This chapter describes the data that should be entered by the user in order to create a proper excavation model.

3.2 Design codes

DeepXcav includes all Eurocode and AASHTO LRFD load specifications. The design approach can be selected from the Analysis tab. There, by pressing the "Single" button, the user can choose to include the factors of a single load case of a specification to the model (Figure 3.2.1). By pressing the "Mult." button, we can choose to generate all load cases of a specification. With this choice, several new design sections are created automatically, including each one a different load case of the selected specification (Figure 3.2.2). Each newly generated design section is automatically linked to its parent design section (all settings except design approach and load combinations)

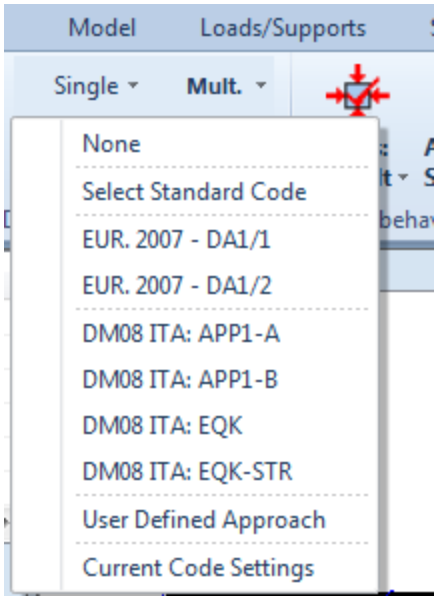


Figure 3.2.1: Single load case

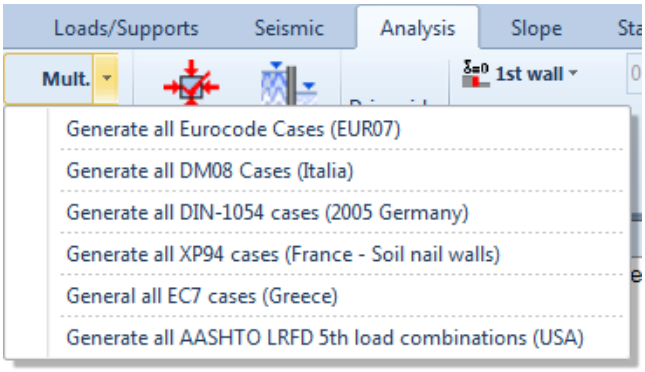


Figure 3.2.2: Multiple load cases

The table below presents the load specifications included in DeepXcav.

Code	Country
DM 08	Italy
DIN - 1054	Germany
XP 94	France
Eurocode 7 Greece	Greece
AASHTO LRFD 5th	USA
Eurocode 7	Europe, general
BS EN-1997-1	British standards DA-1

By selecting the Select Standard Code choice of the Single load case list, the dialog of Figure 3.2.3 appears. In this dialog we can define the design code and specify the load case that should be used in the current design section.

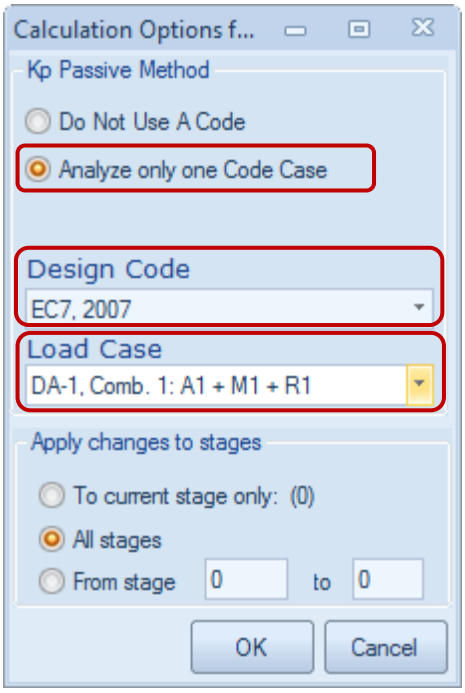


Figure 3.2.3: Select Standard Code dialog.

By selecting to generate all load cases of a specification (i.e. all AASHTO LRFD cases), after the new design sections are automatically created, the user can read the factors used in each load case by pressing the "Mult." button. Then, the Summary of Soil codes dialog appears (Figure 3.2.4). In this dialog, we can manually define load combinations. (Figure 3.2.5)

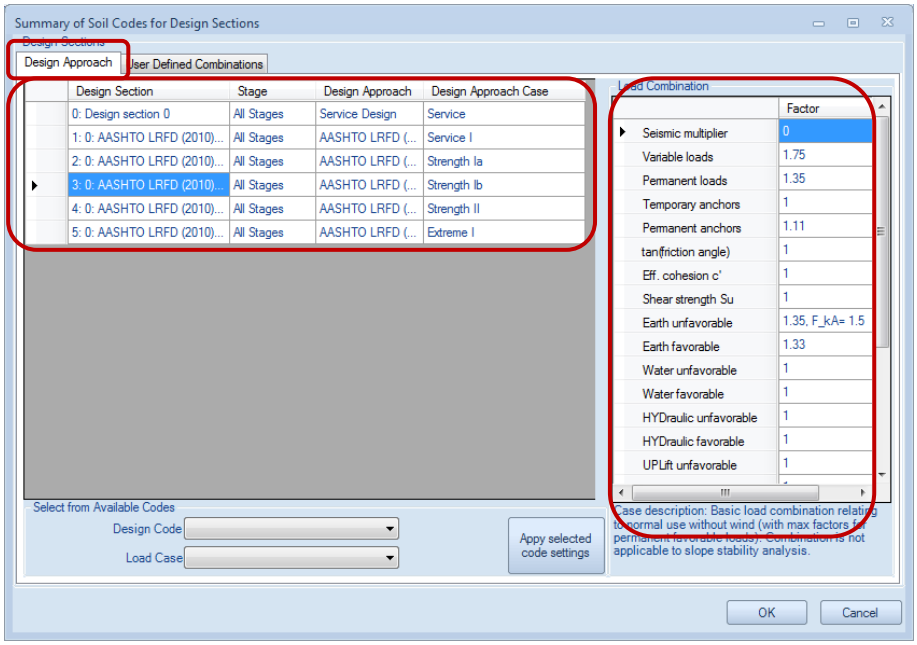


Figure 3.2.4: The Summary of Soil codes dialog.

The table below presents the load combination factors that are contained in the Design approach tab of the summary of soil codes dialog:

Seismic multiplier	Factor for earthquake loads
Variable loads factor	Load factor for variable loads (traffic etc)
Permanent loads factor	Load factor for permanent loads
Temporary anchors factor	Partial resistance factor for temporary anchors (typical design life ≤2 years
Permanent anchors factor	Partial resistance factor for temporary anchors (typical design life ≤2 years
Tan of the friction angle factor	Factor for the soil's friction angle
Effective cohesion factor	Factor for the soil's effective cohesion
Shear strength factor	Partial safety factor on undrained shear strength
Earth favorable factor	Partial safety factor on earth pressures (on the resisting side)
Earth unfavorable factor	Partial safety factor on earth pressures (on the driving side)
Water unfavorable factor	Partial safety factor on water pressures (on the resisting side)
Water favorable factor	Partial safety factor on water pressures (on the driving side)
Hydraulic favorable factor	Partial factor for hydraulic heave checks
Hydraulic unfavorable factor	Partial factor for hydraulic heave checks
Uplift favorable factor	Partial factor for uplift checks
Uplift unfavorable factor	Partial factor for uplift checks
Used structural wall safety factor	Applied safety factor on ultimate wall capacity
Nails qS factor	Partial factor for ultimate bond resistance for soil nails with presumptive values (from charts)

Nails qS (tests) factor	Partial factor for ultimate bond resistance for soil nails with values from actual in-situ tests
Pressuremeter PL factor	Factor for pressure limit of pressuremeter test
Tan of the nail friction angle factor	Factor for the nail's friction angle
Effective cohesion for nail mass factor	Parameter that is used within the soil mass enclosed within soil nails
Undrained shear strength for nail mass factor	Parameter that is used within the soil mass enclosed within soil nails

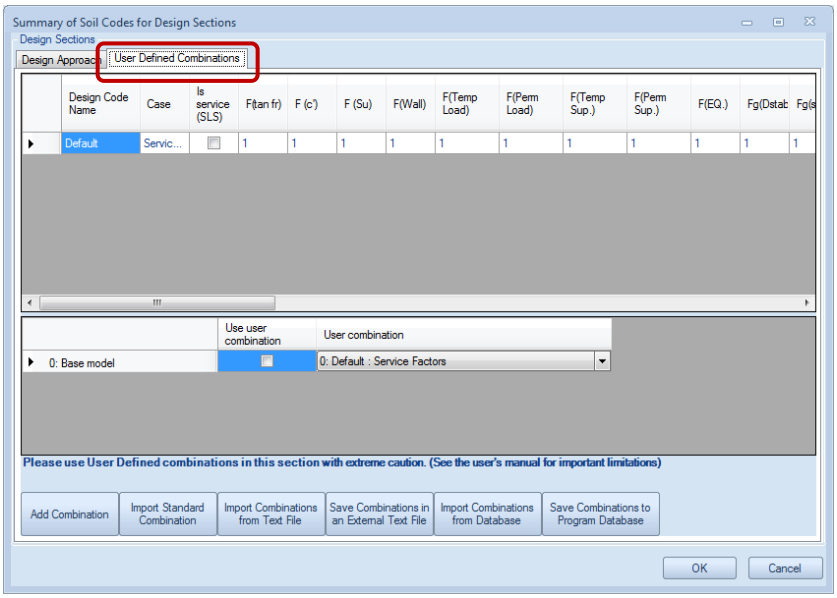


Figure 3.2.5: User defined combinations.

The table below presents the factors that are contained in the user defined combinations tab of the summary of soil codes dialog:

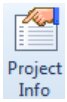
Option to define if the combination Is service (SLS)	This option enables the serviceability checking for concrete cracking
F(tan(fr))	Tan of the friction angle factor
F(c')	Effective cohesion factor
F(Su)	Shear strength factor
F(wall)	Overall safety factor on wall capacity
F(temp load)	Temporary load factor
F(perm load)	Permanent load factor
F(temp sup.)	Temporary support factor
(perm sup.)	Permanent support factor F
Fg(Dstab)	Factor for destabilizing gravity loads
Fg(stab)	Factor for stabilizing gravity loads

F(Earth Res)	Resisting earth factor. This factor is used to divide earth pressures
F(Earth Drive)	Driving earth factor
F(Water Drive)	Driving water factor
F(Water Res)	Resisting water factor
FHyd (stab)	Hydraulic stabilizing factor (used in hydraulic heave calculations)
Hydraulic destabilizing factor FHyd (dStab)	Hydraulic destabilizing factor (used in hydraulic heave calculations)
Option to use multiplication in Paratie	If this option is selected then the initial non-linear analysis is standardized by F(earth drive). The non-linear analysis is then performed and the obtained results are amplified by F(earth drive) to obtain the ultimate design wall moments/shears and support reactions
Option to use F(wall)	Select this option to enable F(wall)
Option to use custom safety factor for qskin	Select this option to use a custom safety factor for ultimate bond resistance for tiebacks
Anchors qskin factor	Partial safety factor for ultimate bond resistance for tiebacks
Soil nails qskin factor	Partial safety factor for ultimate bond resistance for soil nails
Pressuremeter PL factor FS PL(nails)	Pressuremeter PL factor: factor for pressure limit of pressuremeter test for soil nails
Tan of the nail friction angle factor F(Fr nail mass)	Tan of the nail friction angle factor for soil nail mass
Effective cohesion for nail mass factor F(c' nail mass)	Effective cohesion for nail mass factor
F(Su nail mass)	Undrained shear strength for nail mass factor
Use FS_STR_Nails	Option to use structural factor of safety for soil nails
FSstrNail	Structural factor of safety for soil nails
Divide Fpassive at end	With this option the wall embedment GEO checks are performed separately after the basic wall analysis. Hence, the STR and GEO checks in EC7 can be seperated
Use soil nail facing FS	Forces the program to include a structural safety factor for soil nail facings
Soil nail facing permanent safety factor	
Option to use soil reinforcement safety factor*	
Steelstrips Static safety factor*	
Steelstrips Seismic safety factor*	

Steelgrids Static safety factor*	
Steelgrids Seismic safety factor*	
Geosynthetics Static safety factor*	
Geosynthetics Seismic safety factor*	

* The soil reinforcement safety factors are included and refer to a future module.

3.3 Data entry: Project information

By clicking on the button 

, the Project Information dialog appears (Figure 3.3.1). In this dialog we can specify the Project Name, file number (or job number) and the name of the engineer preparing the analysis.

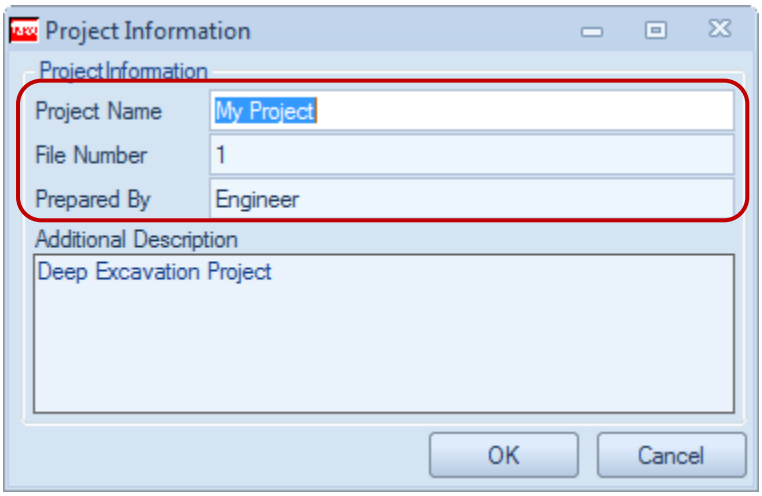


Figure 3.3.1: Project information dialog.

3.4 Data entry: Soil Data

By pressing the Edit soil type data button of the General tab, the Soil Types dialog appears. Here the user can create as many soil types as needed and define their properties. First the user should select to add a new soil type, or choose one from the list in order to modify its properties (Figure 3.4.1).

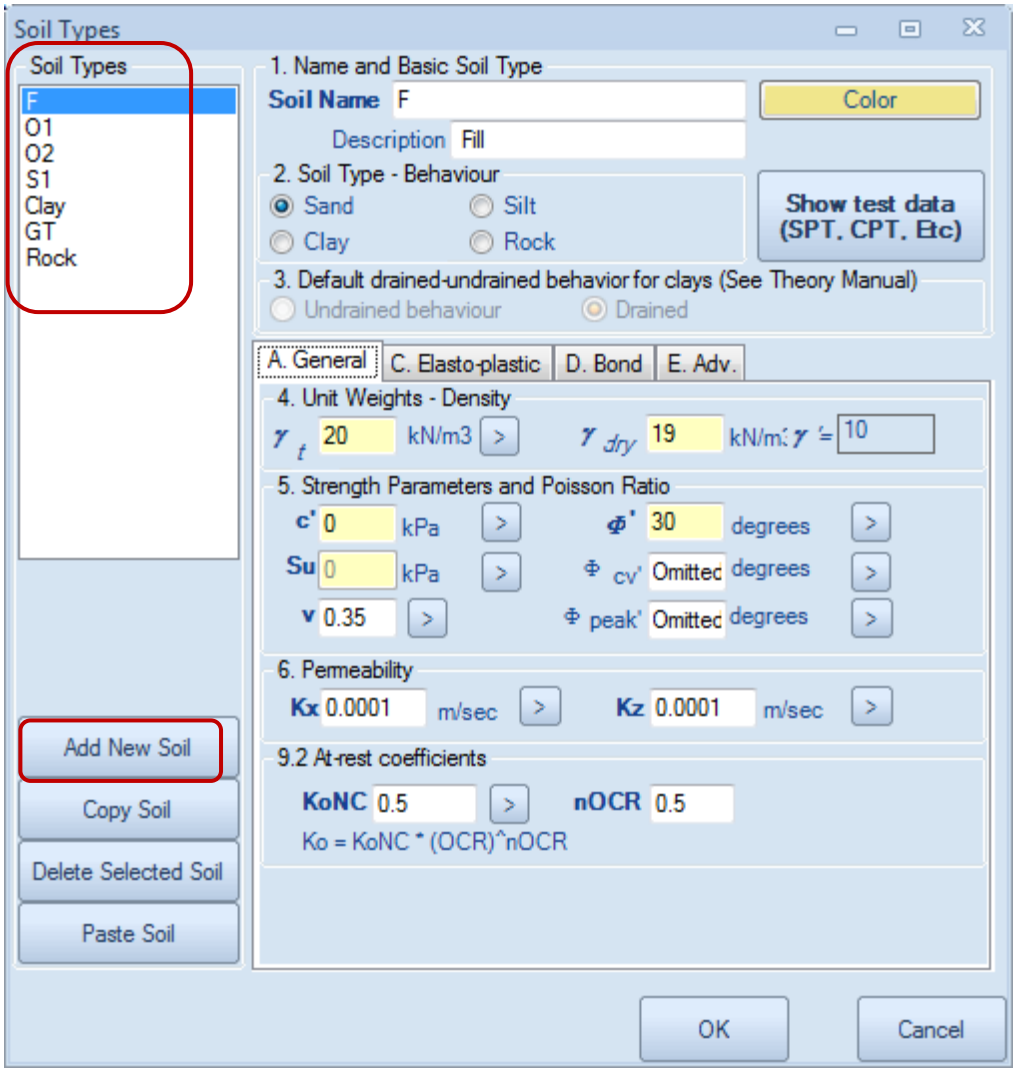


Figure 3.4.1: Add a new soil or select a soil to modify.

We can change the basic soil information such as soil name, preview color and soil type. The available soil types are Sand, Clay, Silt and Rock. If the soil is clay, a drained or undrained behavior of the clay can be defined at this point. All these details can be modified in the area presented in Figure 3.4.2. Next, the user can modify the general soil properties in the area marked in Figure 3.4.3. The basic properties are described in the table below.

Symbol	Description
γ_t	Total unit weight of soil (used below the water table)
γ_{dry}	Dry unit weight of soil (used above the water table)
c'	Effective soil cohesion
Su	Undrained shear strength (used for clays when undrained modeling is selected). In the non-linear analysis this is used as an upper limit strength
v	Poisson's ratio (used for loads calculated with theory of elasticity)
Φ'	Effective soil friction angle
$\Phi_{cv'}$	Constant volume effective shearing soil friction angle used in the non-linear analysis for clays
$\Phi_{peak'}$	Peak effective soil friction angle used in the non-linear analysis for clays
Kx	Soil permeability at horizontal direction
Kz	Soil permeability at vertical direction
KoNC	Coefficient of at-rest lateral earth pressures for normally consolidated conditions
nOCR	Exponent for calculating Ko with Ko=KoNC*[(OCR)^(nOCR)]

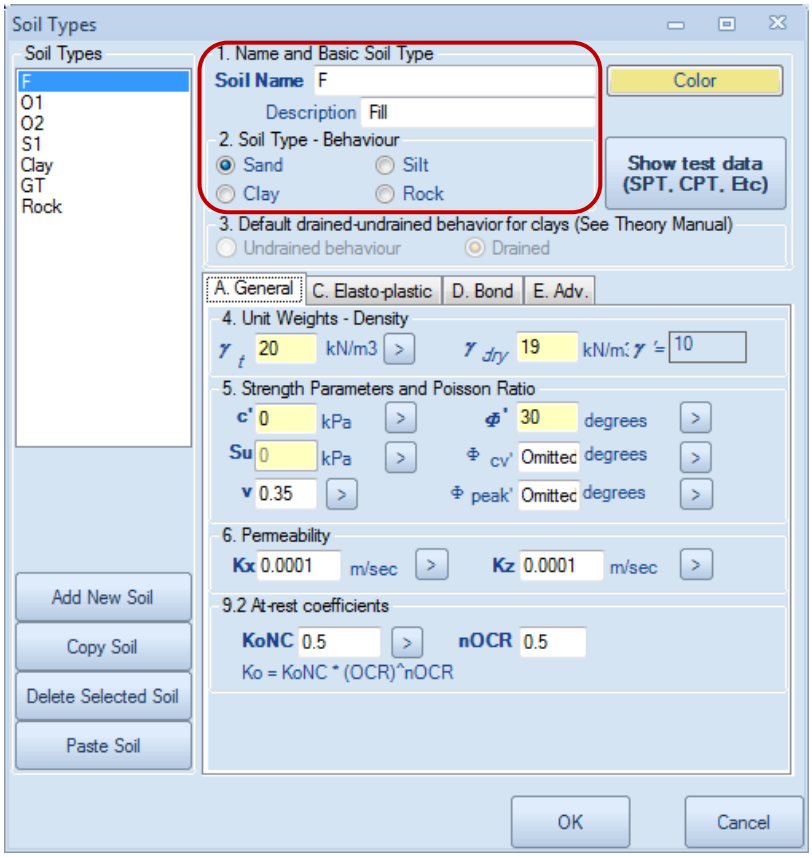


Figure 3.4.2: Define basic soil information.

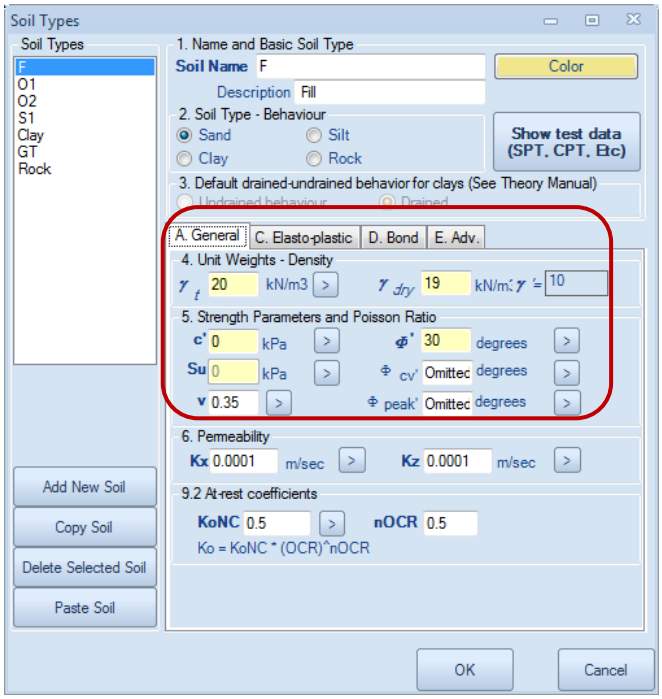


Figure 3.4.3: Define main soil data.

If the user presses the Show Test Data button the dialog expands to the right. Here are available tools for estimating main soil data properties from in-situ tests. Data for standard penetration tests, cone penetrometer tests and pressuremeter tests are included (Figures 3.4.4 and 3.4.5).

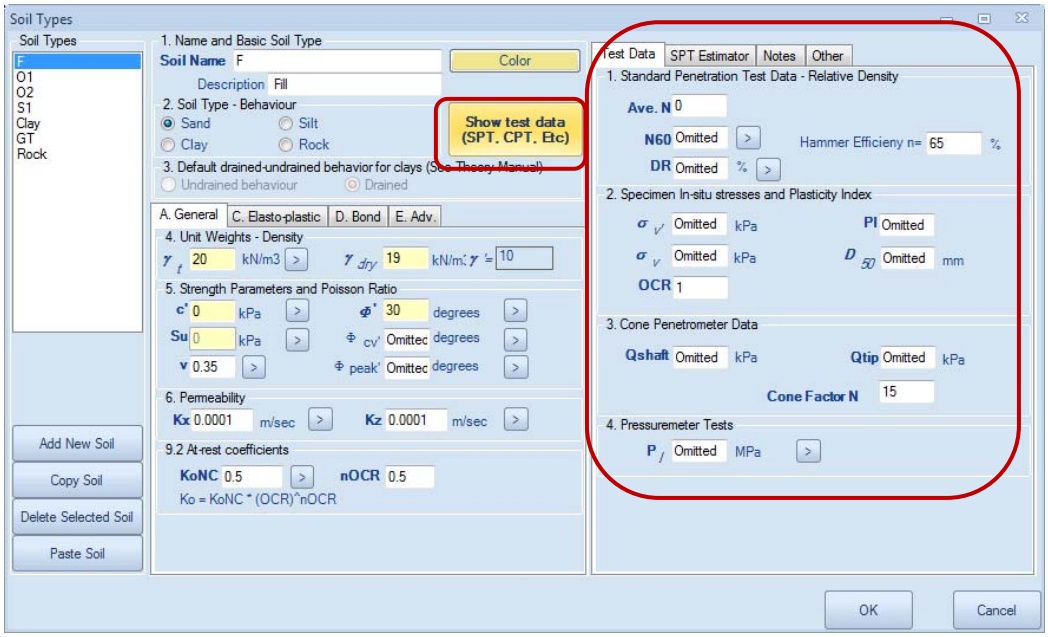


Figure 3.4.4: Test Data.

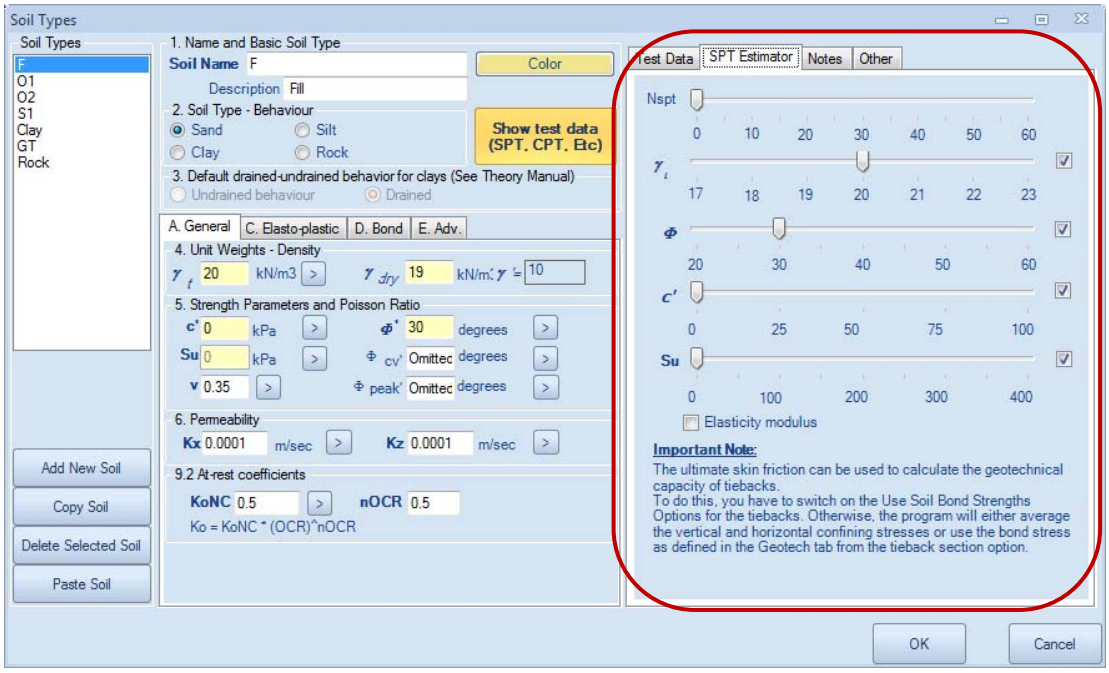


Figure 3.4.5: SPT Estimator.

In addition, next to the main data there is the symbol > . This symbol opens a list of tools that can help the user estimate the soil data values according to published literature.

The tab B.Resistance is initially hidden. It appears when the user selects the User input procedure at the Ka Kp main options dialog (which appears when the user selects the Thrust options choice of the Analysis tab menu, see paragraph 2.9). In this tab, the user can define the initial active and passive coefficients calculation method (Figure 3.4.6). Ka can be calculated by Rankine Equation, or estimated from Graph Correlations (Figure 3.4.7). Kp can be defined either graphically, or by the Rankine or Caquot equations. It is strongly recommended that all initial Ka and Kp values are set to Rankine unless special conditions have to be modeled.

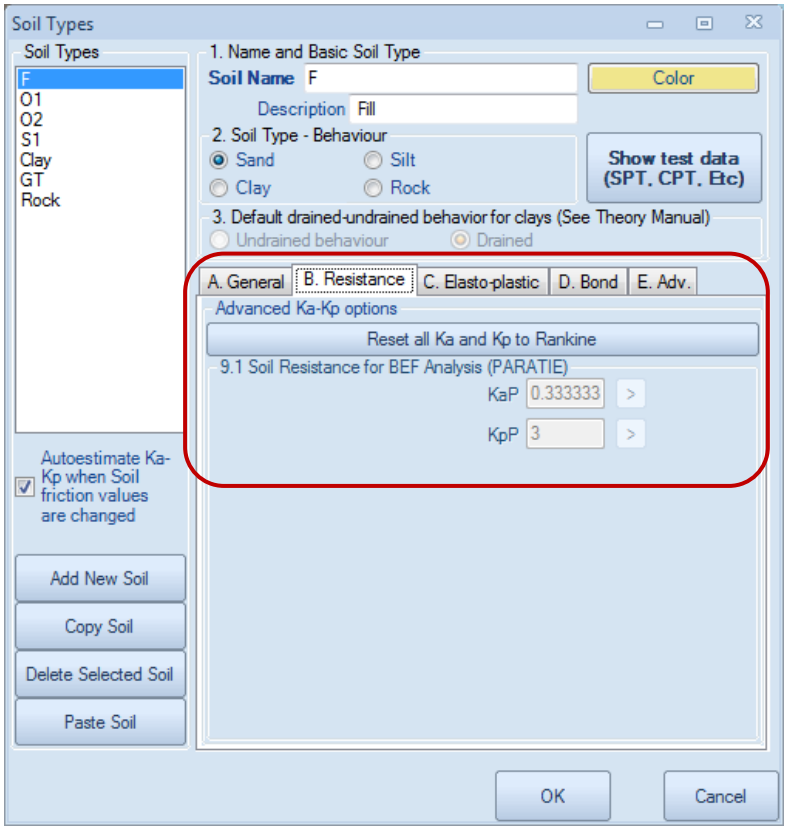


Figure 3.4.6: Ka and Kp calculation.

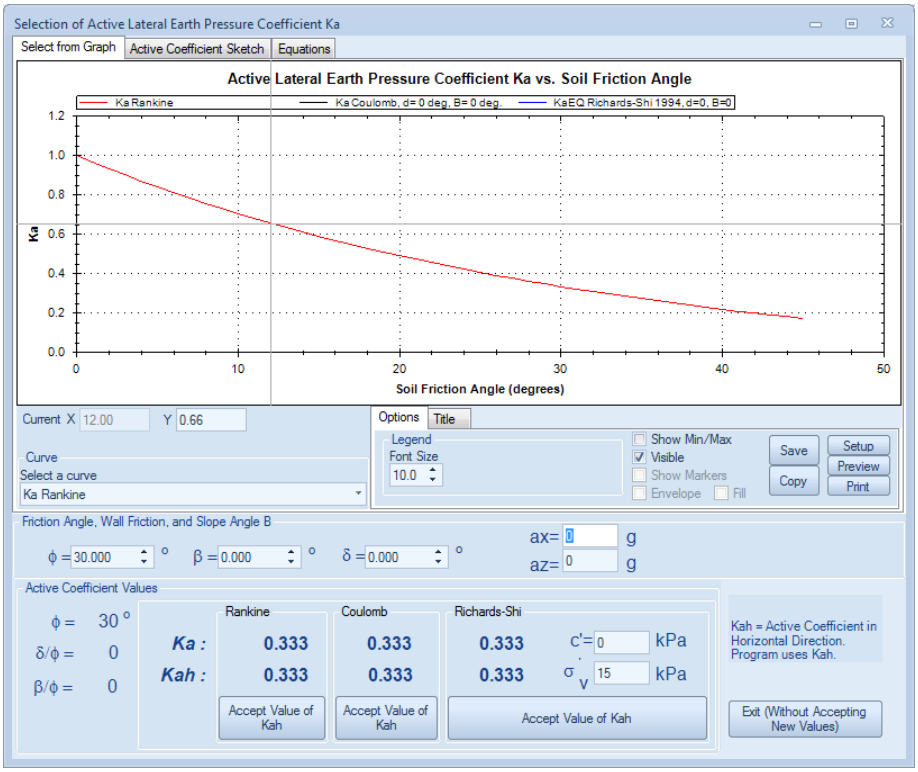


Figure 3.4.7: Graph correlations.

Next, in the tab C we can define the soil elasticity behavior of each soil type (Figure 3.4.8). The elasticity model can be linearly elastic – perfectly plastic, exponential or subgrade-modulus behavior. There are some tools that can help the user estimate loading and reloading elasticity as well as other parameters. In general, the reloading modulus for soils is taken as three times (or greater) than the loading modulus.

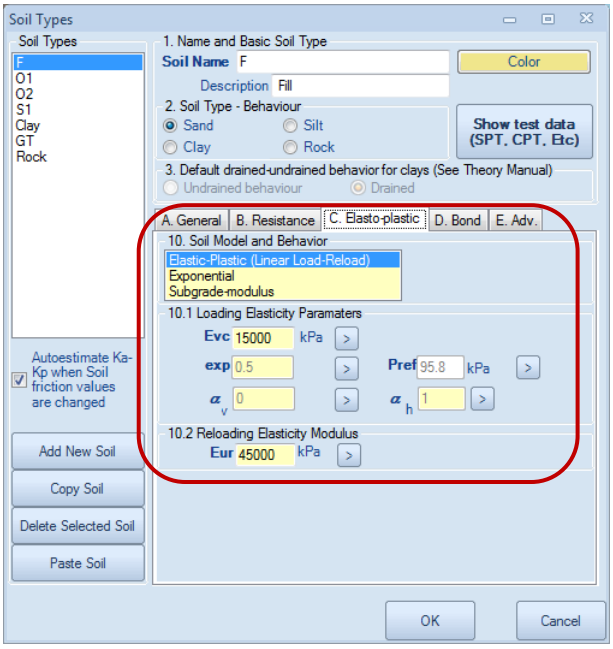


Figure 3.4.8: Loading – reloading elasticity parameters.

The following table presents the properties that are available in the C. Elastoplastic tab:

	Symbol	Description
Elastic-plastic soil behavior	Evc	Virgin compression modulus of elasticity
	Eur	Reloading elasticity modulus
Exponential soil behavior	Eload	Loading elasticity modulus
	exp	Exponent
	av	Coefficient for vertical stress
	ah	Coefficient for horizontal stress
	Pref	Reference pressure
	Eur	Reloading elasticity modulus
Subgrade-modulus soil behavior	Kvc	Loading subgrade reaction modulus
	kur	Reloading subgrade reaction modulus

Finally, if tiebacks or soil nails are going to be used in the model, the user has to define or estimate the Ultimate Bond Resistance.

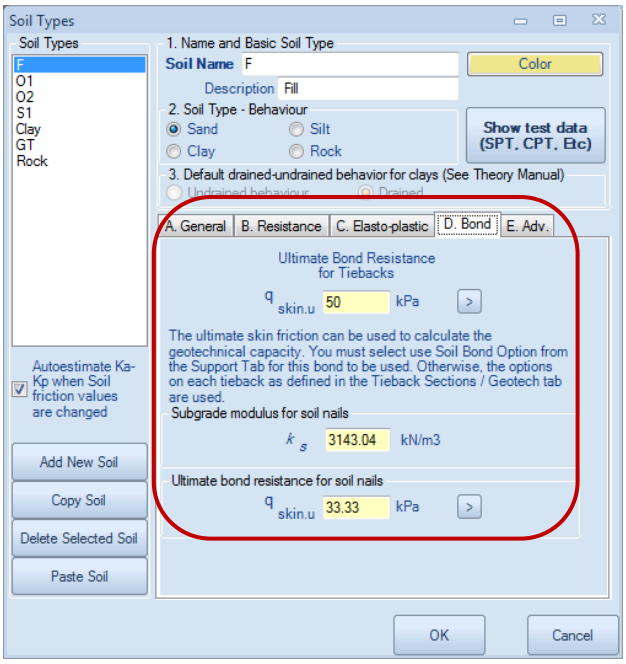


Figure 3.4.9: Ultimate Bond Resistance definition.

The following table presents the properties that are available in the D. Bond tab:

Symbol	Description
q skin,u	Ultimate bond resistance for tiebacks
ks	Subgrade modulus for calculating the shear resistance (in slope stability for soil nails)
q skin,u	Ultimate bond resistance for soil nails

3.5 Data entry: Soil Layers

By pressing the Borings button

of the General tab, the Soil Layers dialog appears. Here we add or remove soil layers, specify their top elevation and the soil type to be used. In addition, we can define the exact position of the boring on the screen. This dialog also appears if the user double-clicks on the boring, or if he selects it from the tree view. All these are presented in Figure 3.5.1. The OCR (overconsolidation ratio) can be defined herein. The user can include as many borings as desired. Each independent design section can include a different boring.

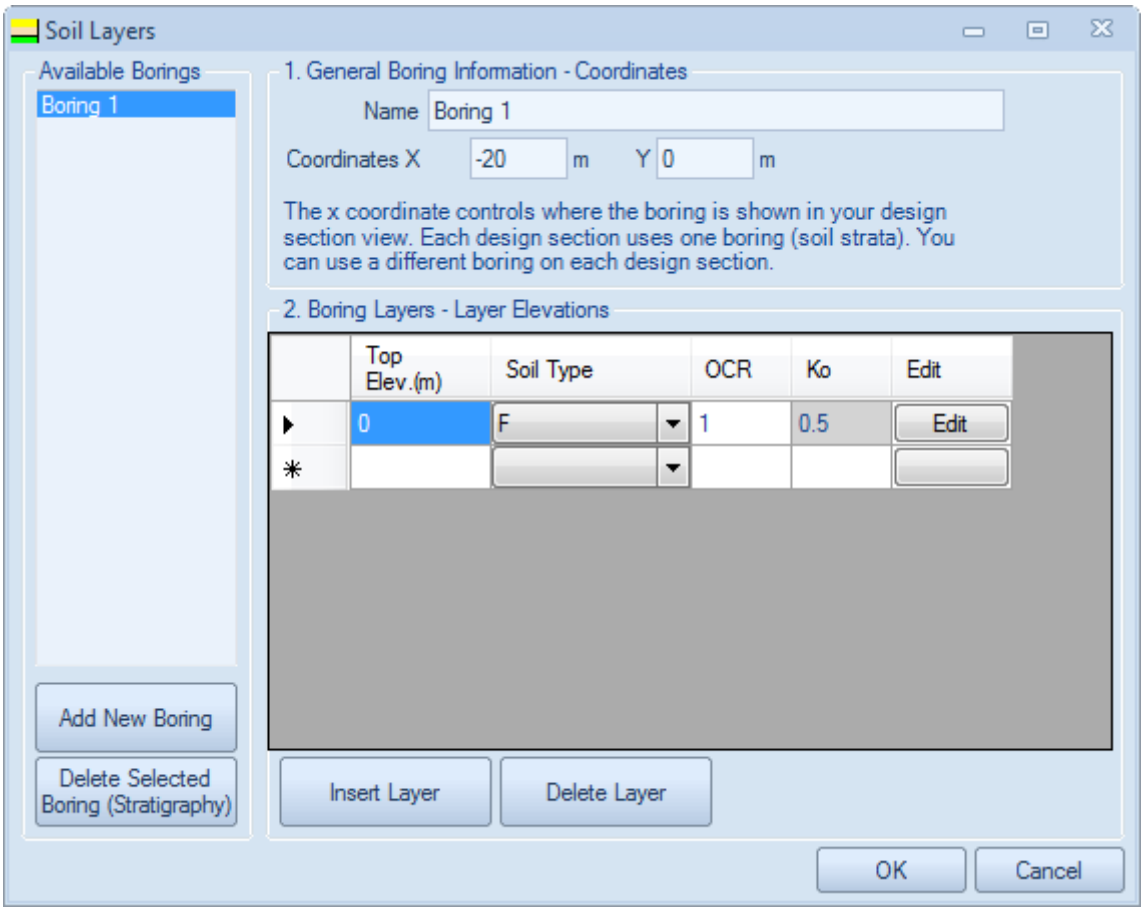


Figure 3.5.1: The soil layers dialog.

3.6 Data entry: Water

In the Analysis tab, the user can select the water analysis method. DeepXcav contains the following water analysis methods:

- Hydrostatic
- Simplified flow
- Full flownet
- User pressures

The option Seal Excavation (create liner) seals the bottom of the excavation against water flow.

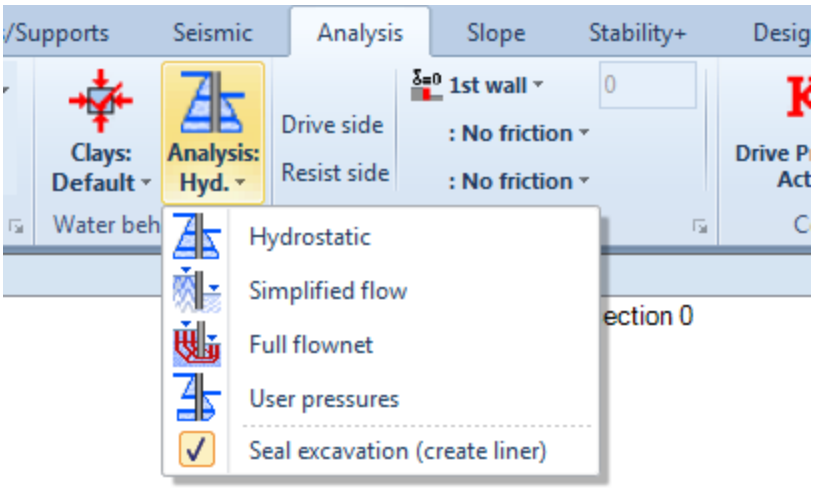


Figure 3.6.1: Water options.

The user is asked after choosing one of the methods above whether to apply this to all construction stages or to apply it only to the current stage. **All these options have been presented in paragraph 2.9.**

3.7 Data entry: Wall data

In DeepXcav, each wall within the model uses a wall section. A wall section can be used in many walls across many design sections. By double clicking on a wall or by choosing a wall from the tree view, the Edit wall data dialog appears. Here we can define the basic wall properties such as wall length, top wall elevation, wall coordinates and number of nodes used in LEM calculations. In addition, by clicking on the Edit section data button, the user can change and modify the wall section used (Figures 3.7.1, 3.7.2). We can also select which wall section to use. The element-behaves-as-slave option controls if the wall is connected to the slave nodes (advanced option). For the beam continuity it is recommended to keep all options selected.

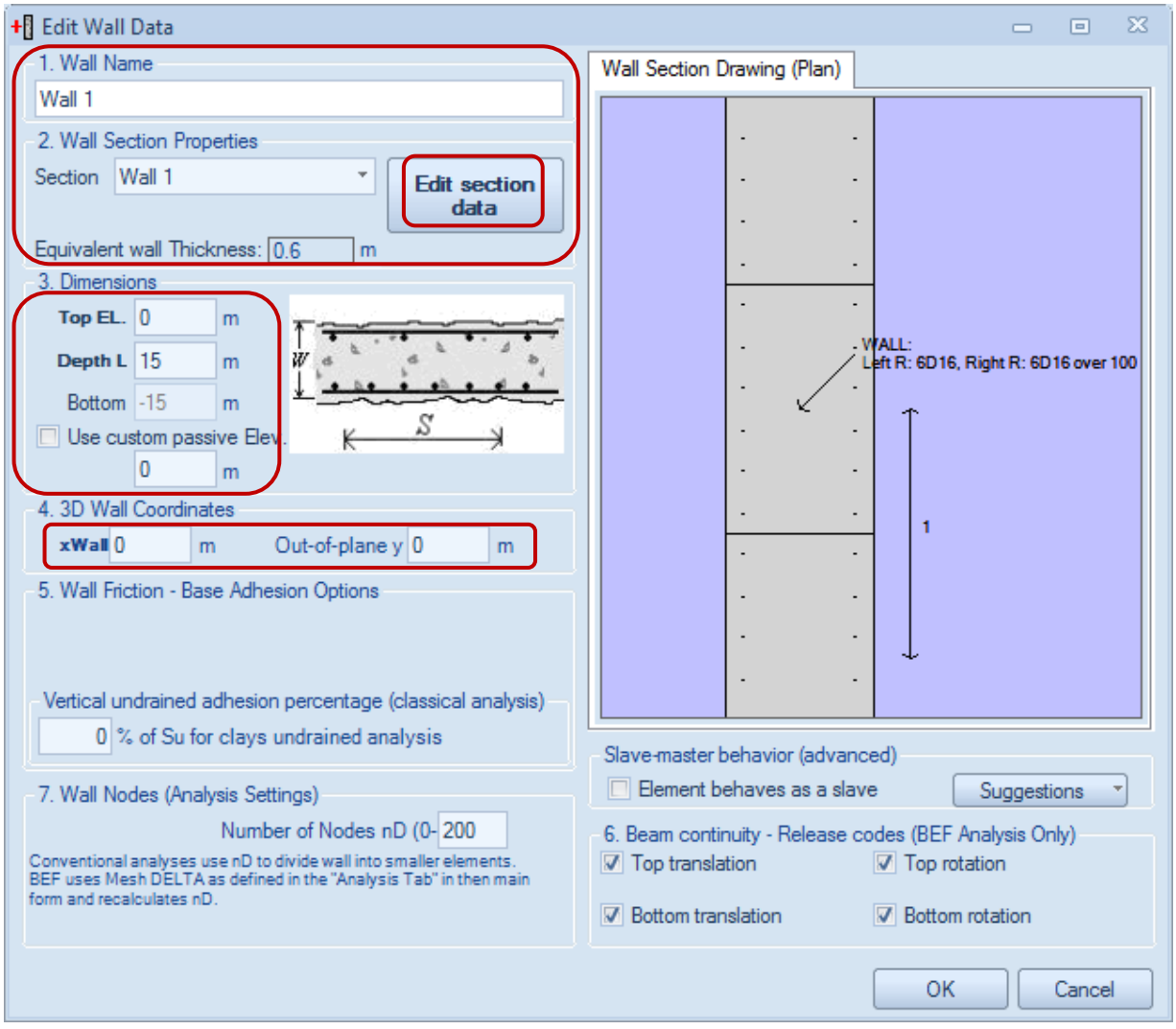


Figure 3.7.1: Edit wall data dialog.

3.7.1 Data entry: Wall sections

Wall sections are databases of wall properties that can be used in any wall within a design section.

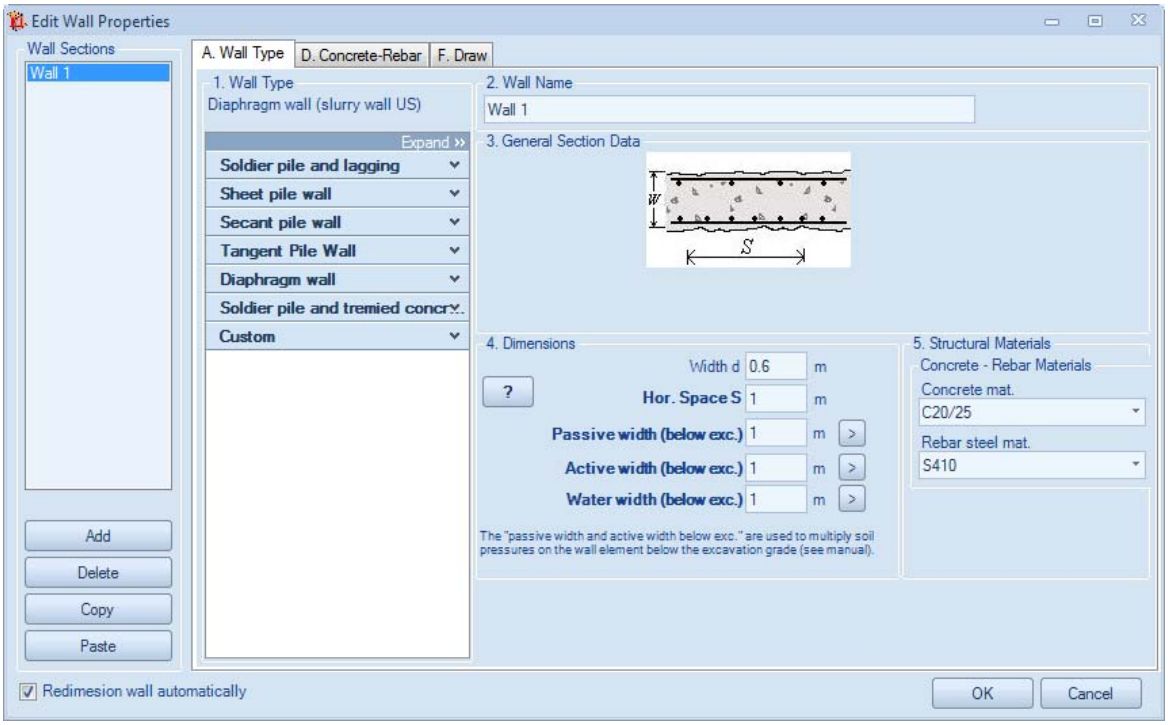
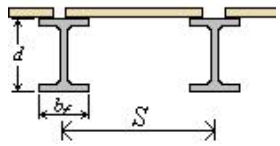
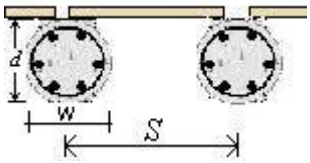
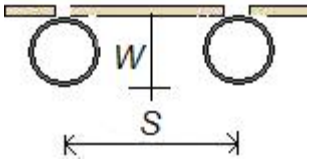
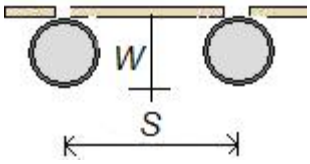
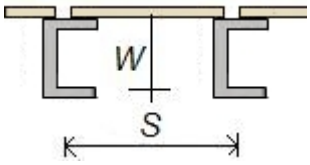
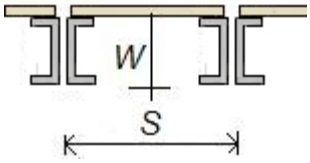
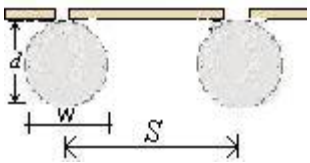


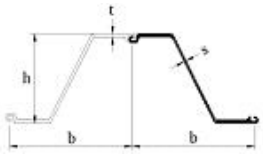
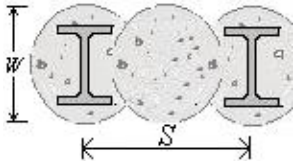
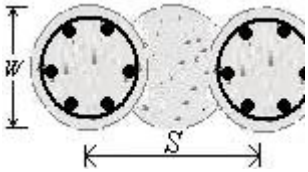
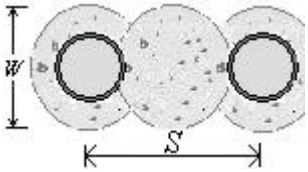
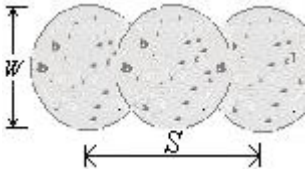
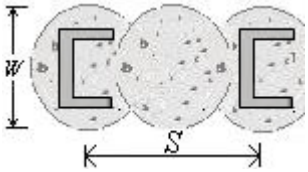
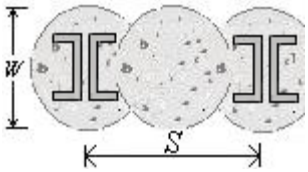
Figure 3.7.2.a: Edit wall properties dialog.

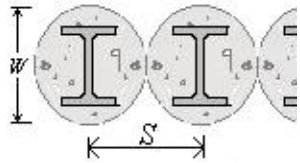
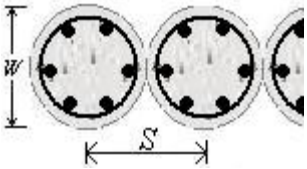
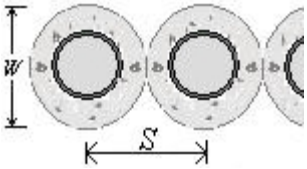
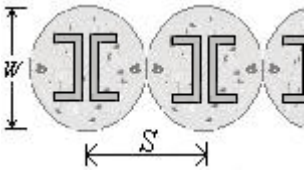
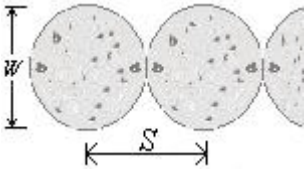
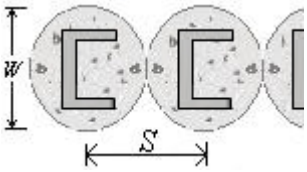
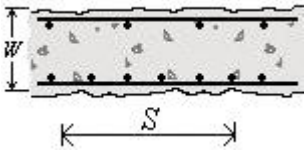
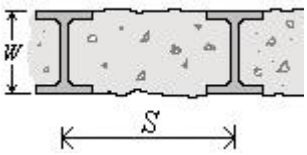
The following table presents the dimension properties that are available in the wall sections dialog:

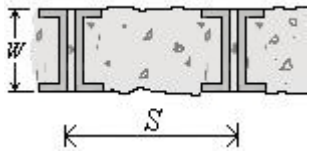
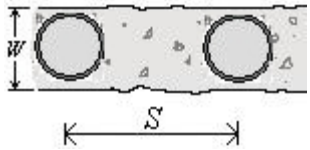
Symbol	Description
d	The width of the wall
S	The horizontal spacing of the wall
Passive width	Passive horizontal tributary width
Active width	Active horizontal tributary width
Water width	Water horizontal tributary width

DeepXcav contains the wall types that are presented in the table below. The data entry of each wall type will be described in the following paragraphs.

Option	Description
	Select this option to use a soldier pile and lagging wall, supported by H-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and lagging wall, supported by reinforced concrete beams. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and lagging wall, supported by pipe-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and lagging wall, supported by pipe-section steel beams, filled with concrete. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and lagging wall, supported by channel-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and lagging wall, supported by double channel-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and lagging wall, supported by plain concrete beams. This option will deactivate all other wall type parameters.

Wall Type - Sheet Piles 	Select this option to use a sheet pile wall. This option will deactivate all other wall type parameters.
	Select this option to use a secant pile wall, supported by H-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a secant pile wall, supported by reinforced concrete piles. This option will deactivate all other wall type parameters.
	Select this option to use a secant pile wall, supported by pipe-section steel beams, filled with concrete. This option will deactivate all other wall type parameters.
	Select this option to use a secant pile wall. This option will deactivate all other wall type parameters.
	Select this option to use a secant pile wall, supported by channel-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a secant pile wall, supported by double channel-section steel beams. This option will deactivate all other wall type parameters.

	Select this option to use a tangent pile wall, supported by H-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a tangent pile wall, supported by reinforced concrete piles. This option will deactivate all other wall type parameters.
	Select this option to use a tangent pile wall, supported by -section steel beams, filled with concrete. This option will deactivate all other wall type parameters.
	Select this option to use a tangent pile wall, supported by double channel-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a tangent pile wall. This option will deactivate all other wall type parameters.
	Select this option to use a tangent pile wall, supported by channel-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use diaphragm wall. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and tremied concrete wall, supported by H-section steel beams. This option will deactivate all other wall type parameters.

	Select this option to use a soldier pile and tremied concrete wall, supported by double channel-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and tremied concrete wall, supported by -section steel beams, filled with concrete. This option will deactivate all other wall type parameters.
	Select this option to use a soldier pile and tremied concrete wall, supported by channel-section steel beams. This option will deactivate all other wall type parameters.
	Select this option to create a wall with manually defined properties. This option will deactivate all other wall type parameters.

By clicking on the button ? of the wall sections dialog, the dialog in Figure 3.7.2.b appears. This dialog contains useful information regarding the passive and active widths.

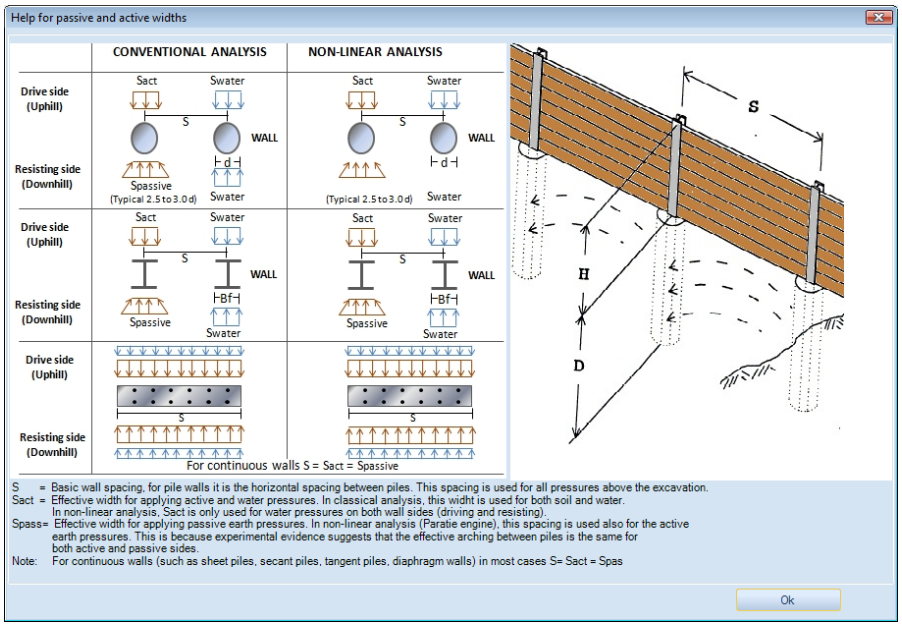

Help for passive and active widths
CONVENTIONAL ANALYSIS
NON-LINEAR ANALYSIS
Drive side (Uphill)
Resisting side (Downhill)
Sact
Swater
Spassive
(Typical 2.5 to 3.0 d)
WALL
Bf
d
S
H
D
For continuous walls S = Sact = Spassive
S = Basic wall spacing, for pile walls it is the horizontal spacing between piles. This spacing is used for all pressures above the excavation.
Sact = Effective width for applying active and water pressures. In classical analysis, this widht is used for both soil and water. In non-linear analysis, Sact is only used for water pressures on both wall sides (driving and resisting).
Spass= Effective width for applying passive earth pressures. In non-linear analysis (Paratie engine), this spacing is used also for the active earth pressures. This is because experimental evidence suggests that the effective arching between piles is the same for both active and passive sides.
Note: For continuous walls (such as sheet piles, secant piles, tangent piles, diaphragm walls) in most cases S= Sact = Spas
Ok

Figure 3.7.2.b: Help for passive and active widths dialog.

3.7.2 Wall type: Soldier piles

At the wall sections dialog, we user can choose to use a soldier pile wall. The user can choose to use a standard steel section from the area marked in Figure 3.7.3. DeepXcav includes all European and US standard sections. In this dialog can modify the dimensions of the wall.

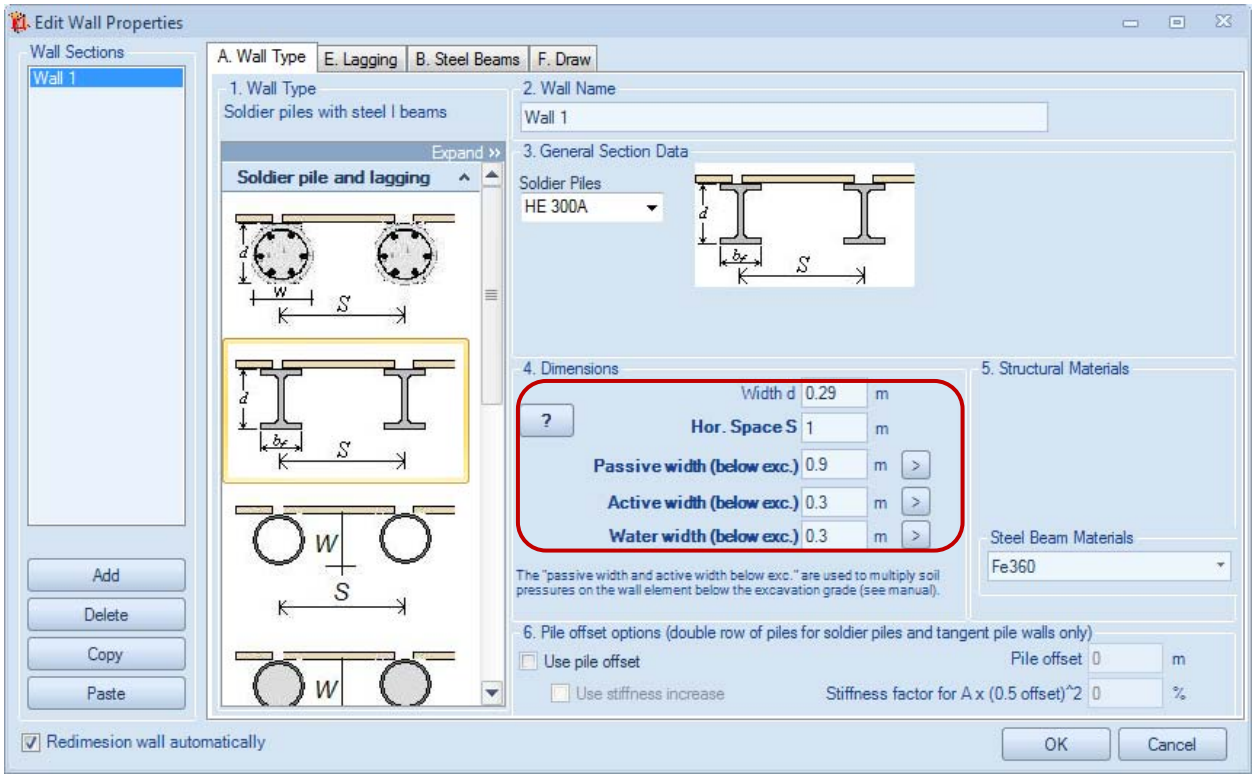


Figure 3.7.3: General section and dimensions for Soldier pile walls.

In E. Lagging tab (Figure 3.7.4), we can modify the lagging properties, such us the lagging type (timber or concrete), and the lagging thickness and section. Finally, here we can also define the lagging position in the wall.

In B. Steel Beams tab we can define the steel beam properties or select a standard section. The section can either be a normal H-section (Figures 3.7.5 and 3.7.6), a single or double channel section (Figures 3.7.7 and 3.7.8) or a Pipe section, filled or not with concrete (Figures 3.7.9 and 3.7.10).

Some other useful options like Reduce steel pipe capacity due to threads and Effectiveness of concrete for stiffness calculations are available in this dialog (Figure 3.7.11). Finally, the user can manually set the section properties my choosing the Edit strut properties manually option (Figure 3.7.12).

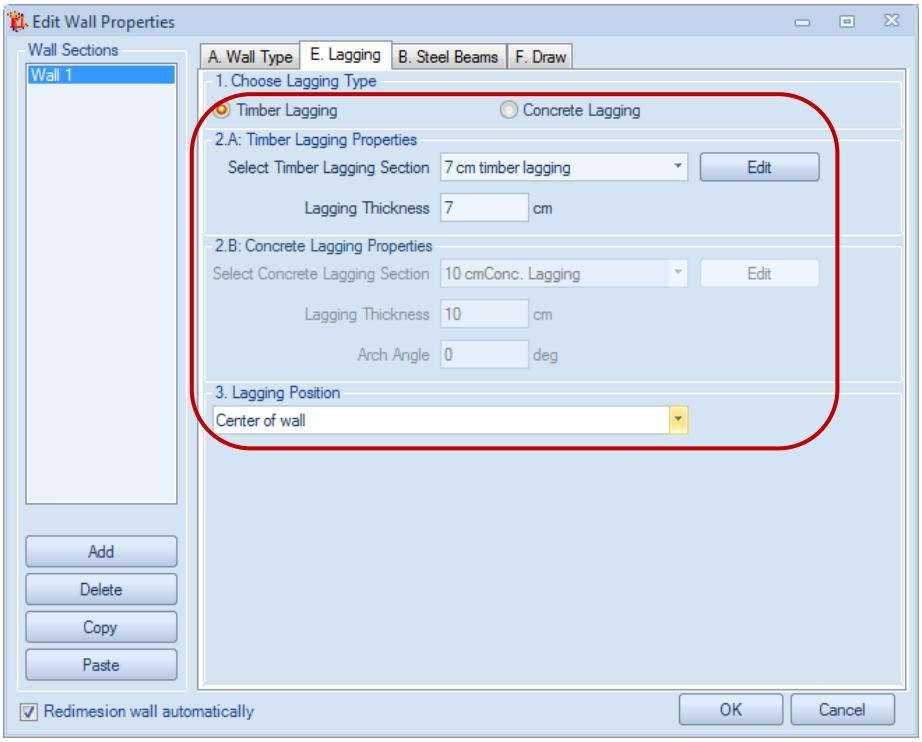


Figure 3.7.4: Lagging properties.

The reduce steel pipe capacity due to threads option reduces the section modulus of a steel pipe to account for strength reductions due to the presence of threads in the piping. Fully welded pipes will likely provide all the theoretical capacity and will likely not require that this feature is activated.

The available section properties are presented and described in the table below.

Symbol	Description
D	Depth or diameter
A	Area
Ixx	Strong axis moment of inertia
Iyy	Weak axis moment of inertia
Sxx	Strong axis elastic section modulus
Syy	Weak axis elastic section modulus
rx	Strong axis radius of gyration
ry	Weak axis radius of gyration
Zxx	Strong axis plastic section modulus
Zyy	Weak axis plastic section modulus
tP	Thickness
J	Rotational moment of inertia
W	Wrapping constant

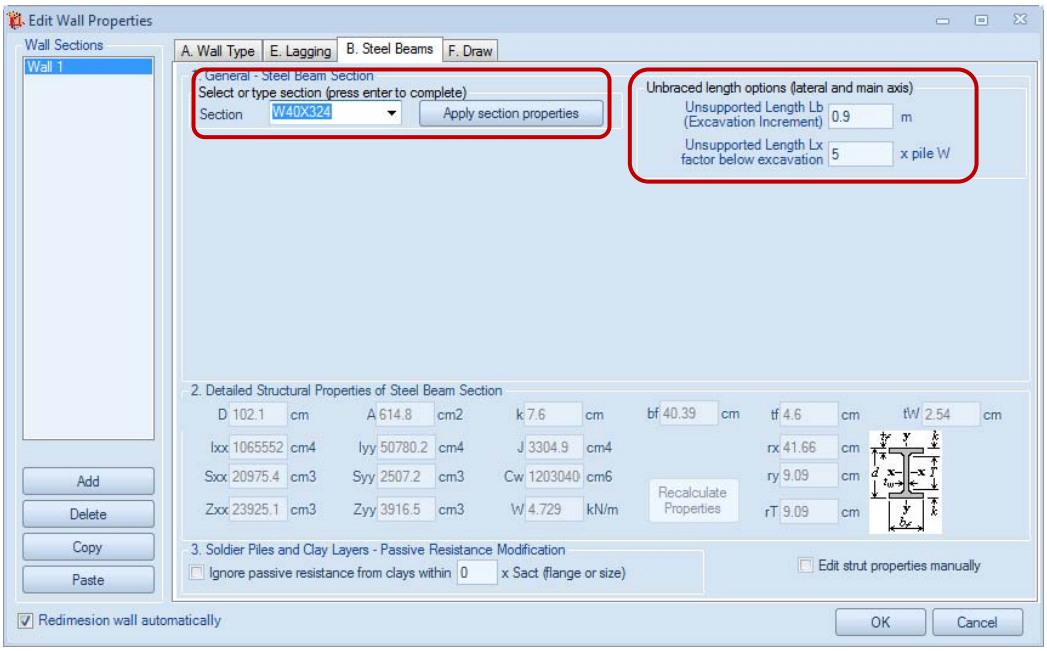


Figure 3.7.5: T-sections.

The parameters Unsupported length Lb and unsupported length Lx factor below excavation control the buckling calculations for the wall capacity.

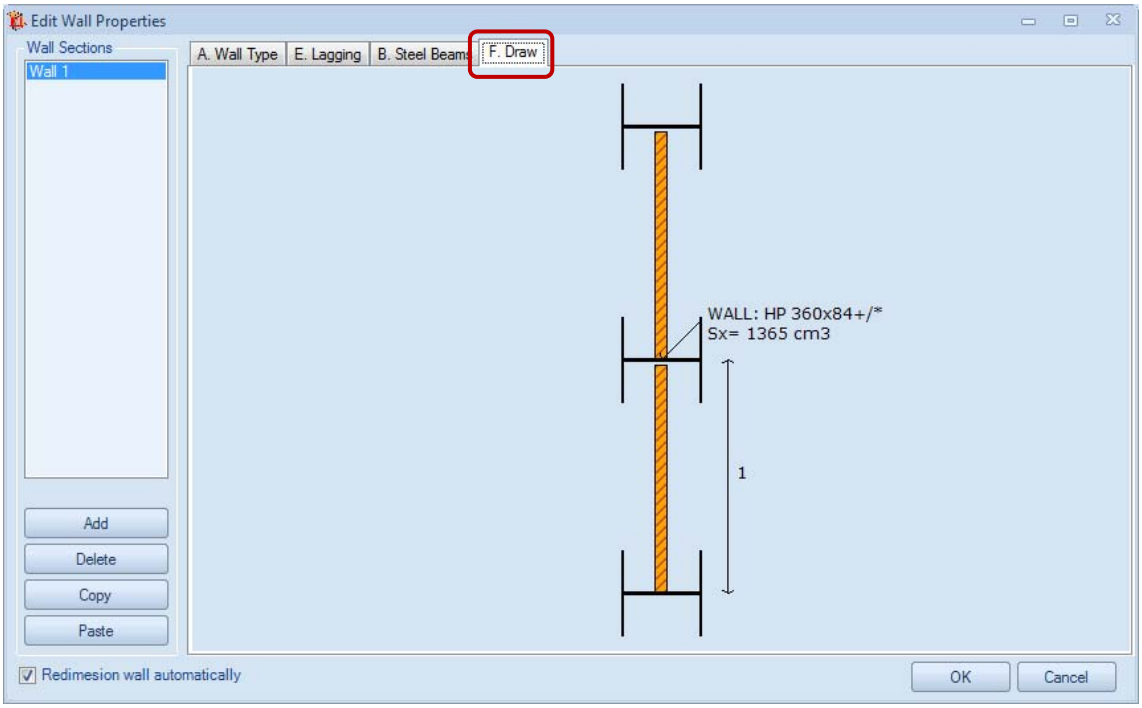


Figure 3.7.6: H-section soldier pile wall plan.

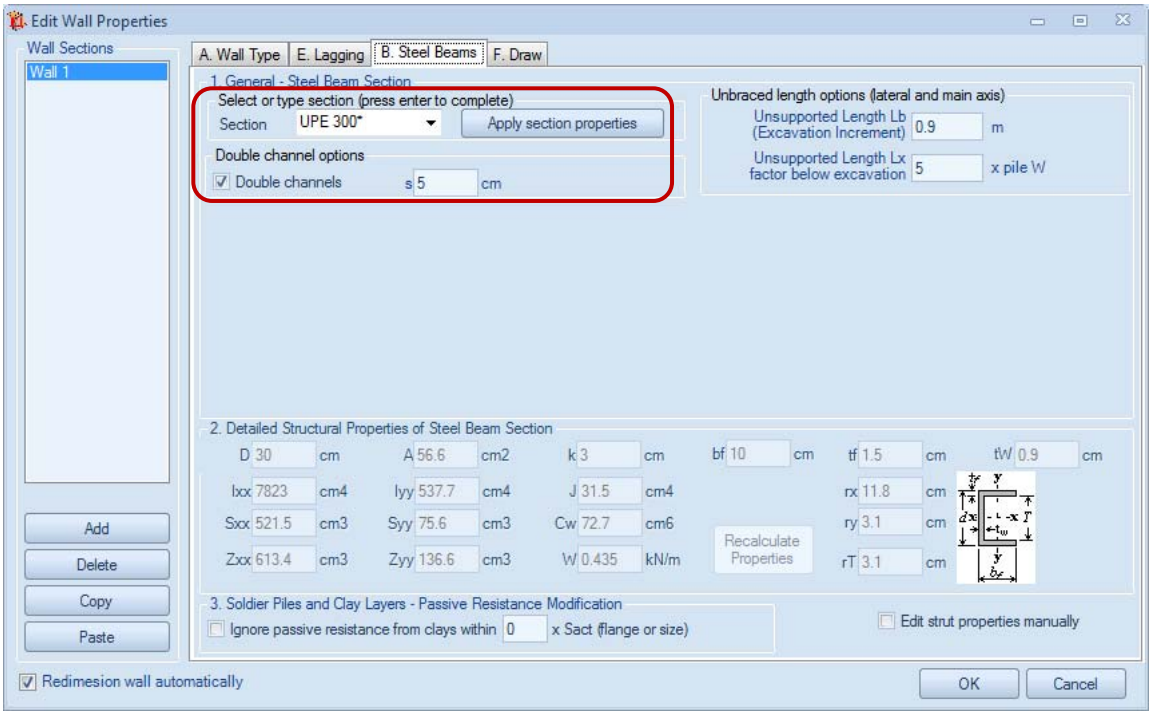


Figure 3.7.7: Double channel sections.

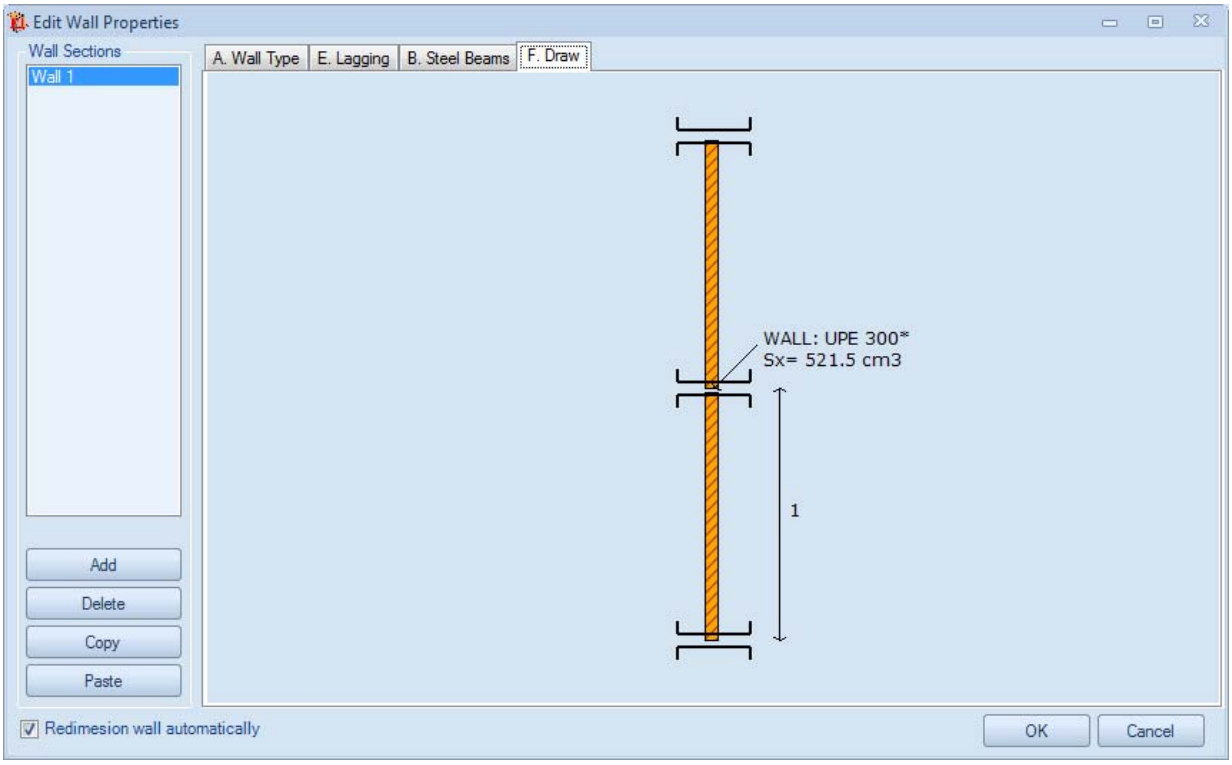


Figure 3.7.8: Double channel section soldier pile wall plan.

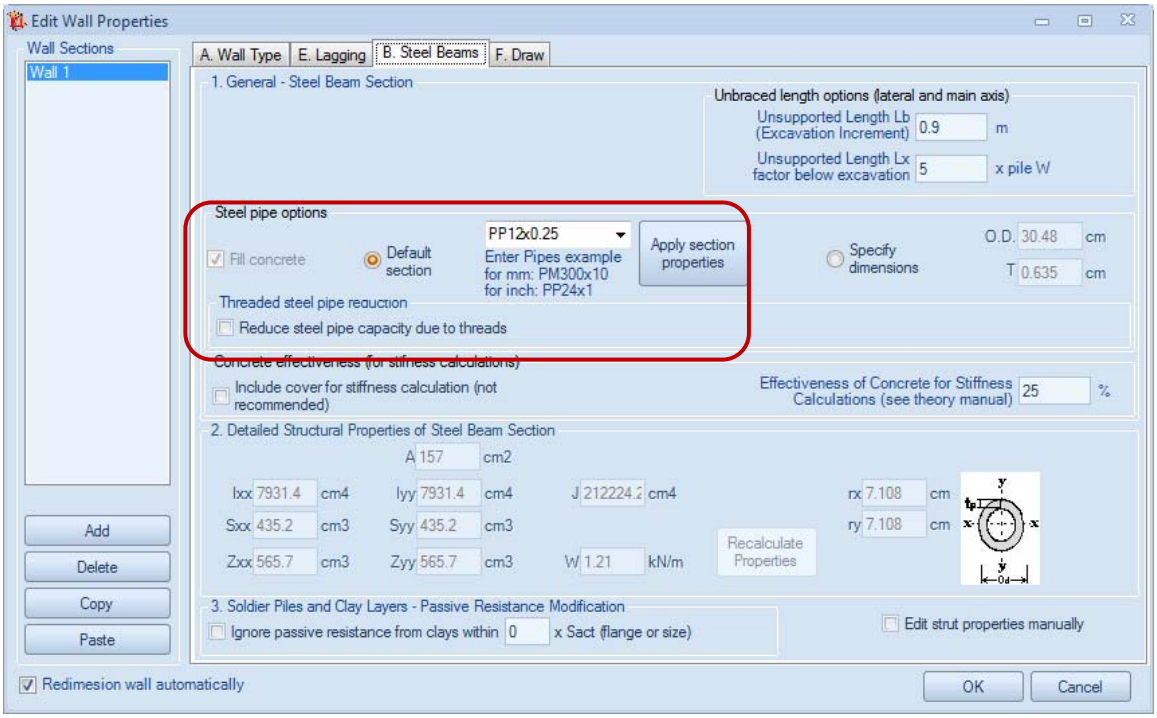


Figure 3.7.9: Pipe sections filled with concrete.

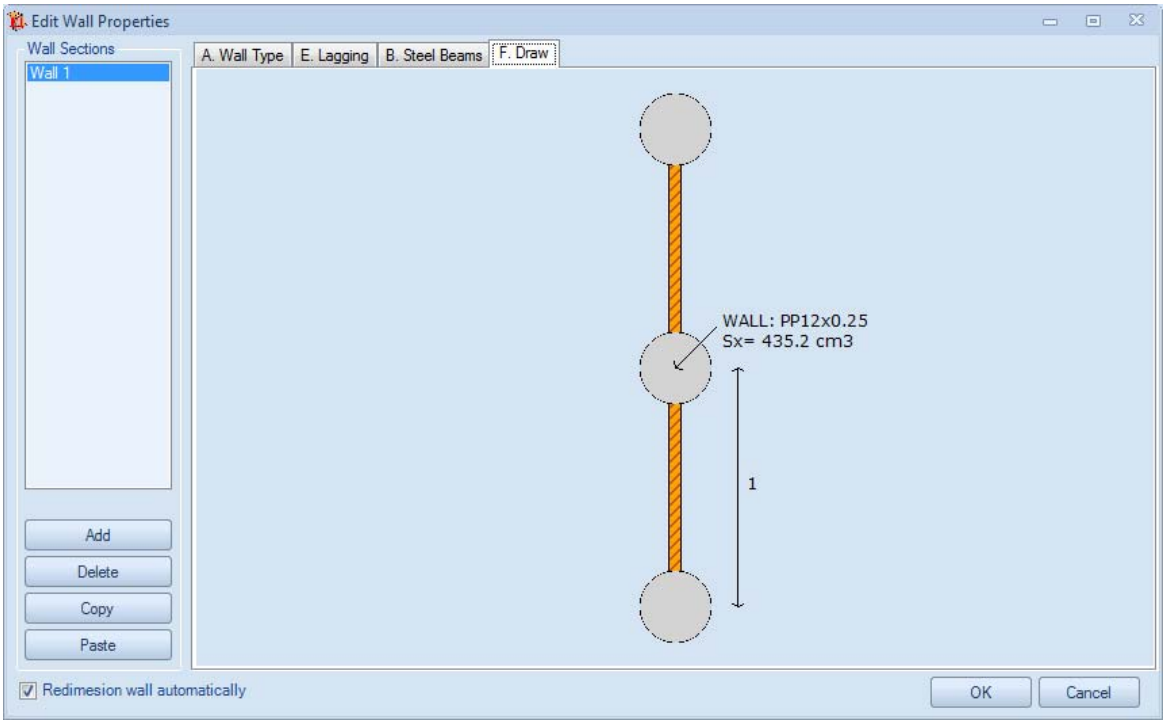


Figure 3.7.10: Pipe section filled with concrete soldier pile wall plan.

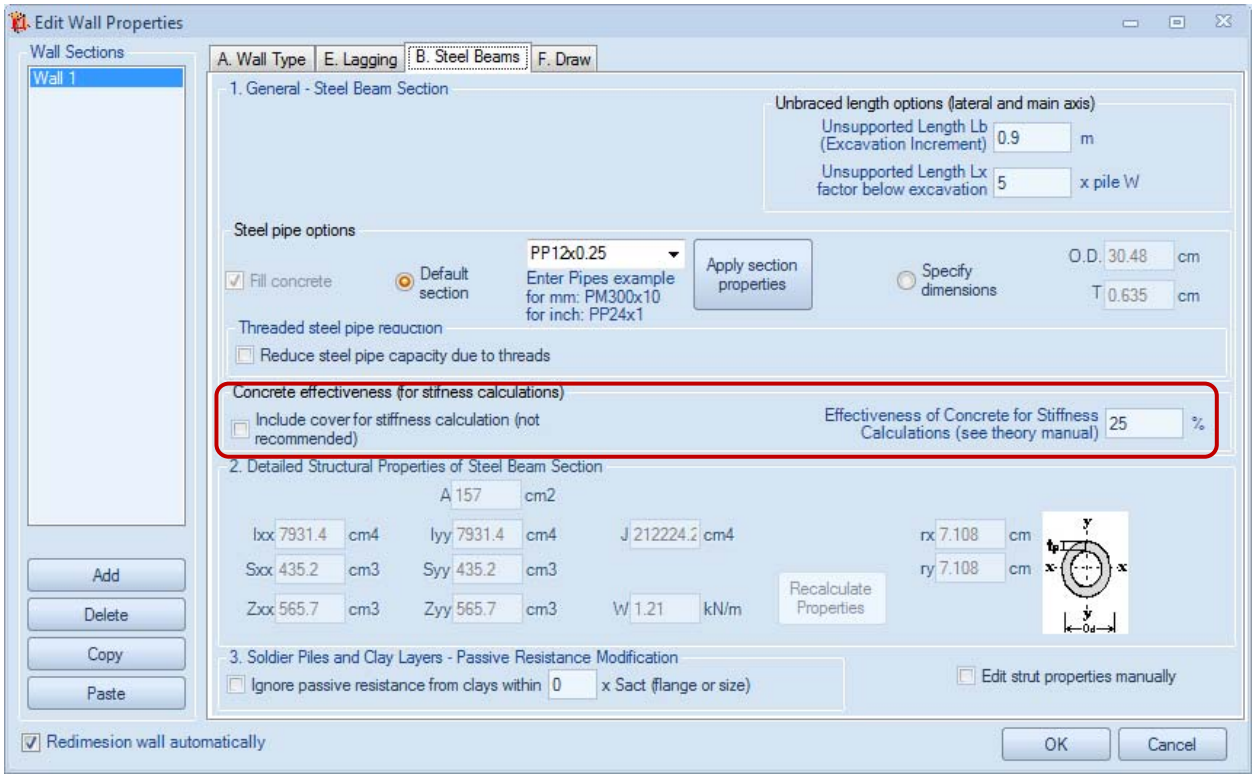


Figure 7.3.11: Other options.

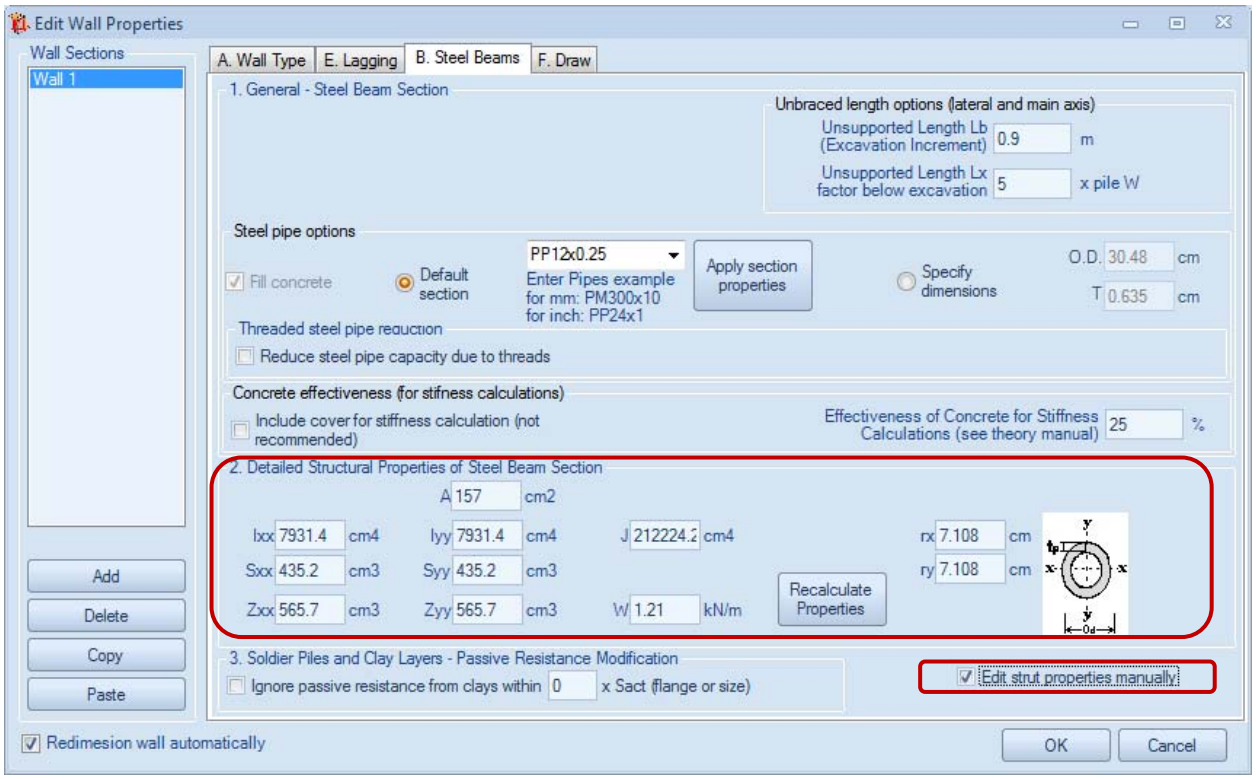


Figure 7.3.12: Edit sections manually.

3.7.3 Wall type: Sheet piles

At the wall sections dialog, we can select to use sheet pile walls. The user can choose to use a standard sheet pile section from the area marked in Figure 3.7.13.

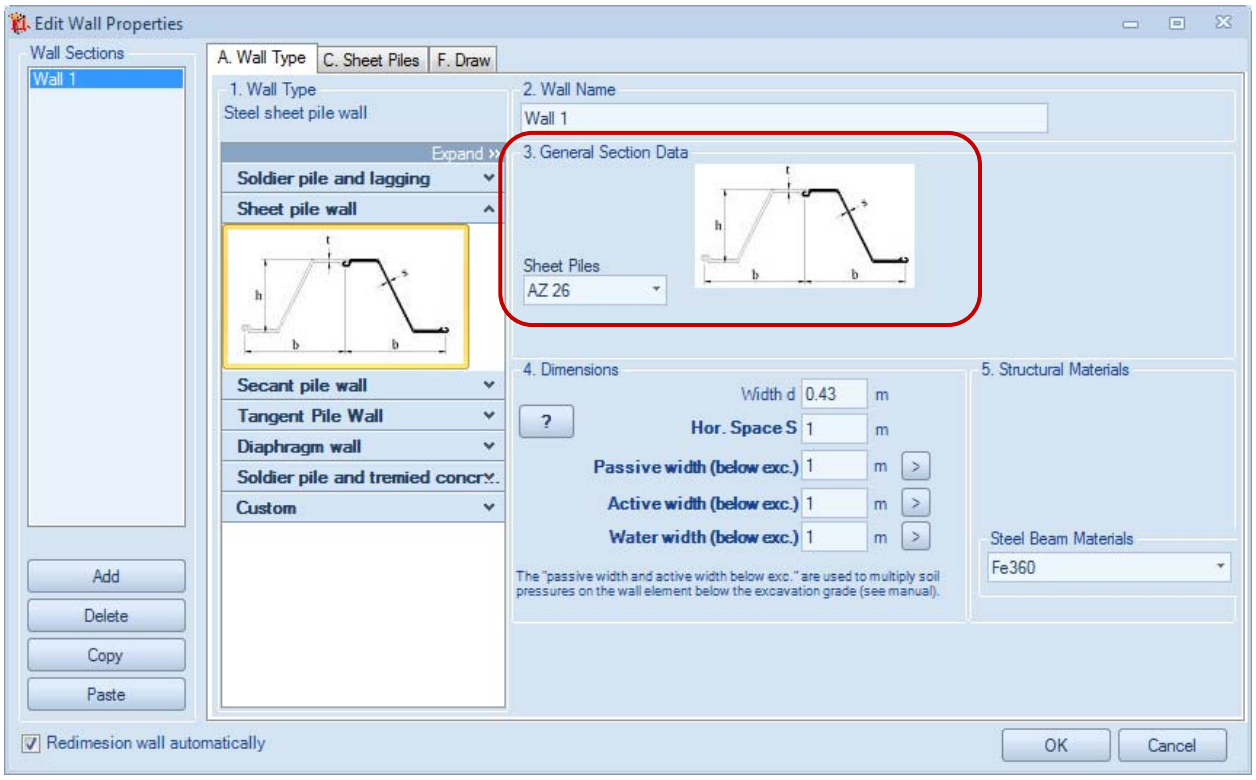


Figure 3.7.13: General section and dimensions for sheet pile walls.

In C. sheet piles tab (Figure 3.7.14), we can change the sheet pile properties. These properties are presented in the following table. Figure 3.7.15 shows the plan of a sheet pile wall as presented in F. Draw tab.

Symbol	Description
h	Height
A	Area
b	Base length
tf	Flange thickness
Ixx	Strong axis moment of inertia
Sxx	Strong axis elastic section modulus
s	Web thickness
α	Base angle of web from horizontal axis

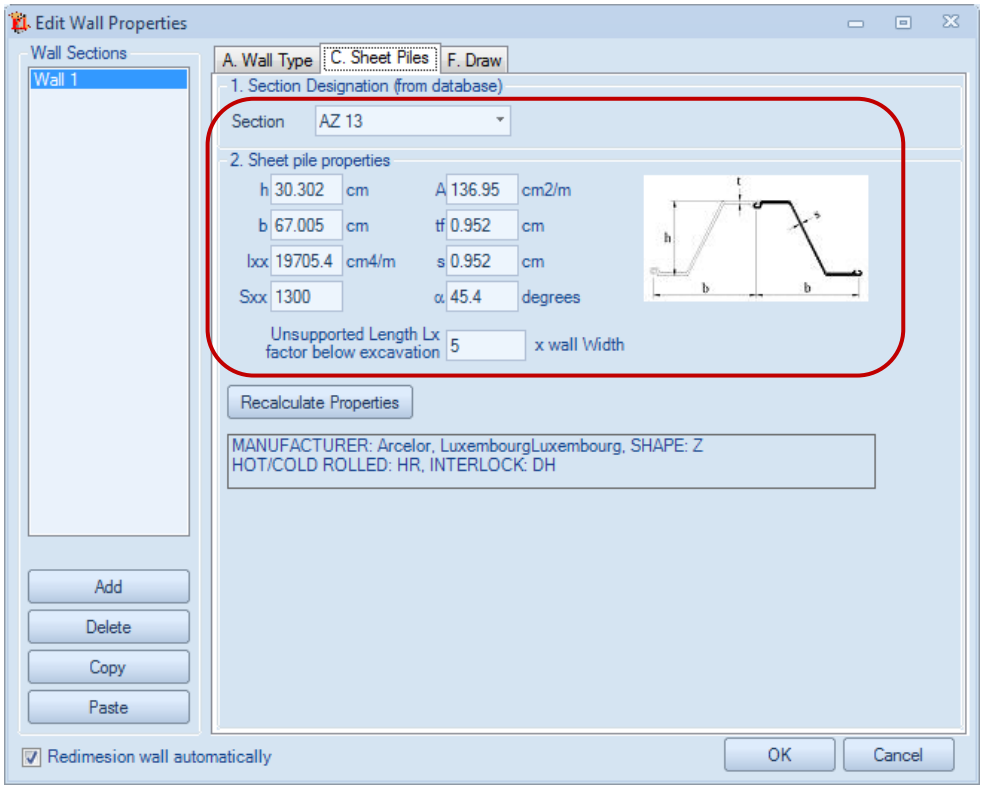


Figure 3.7.14: Sheet pile properties.

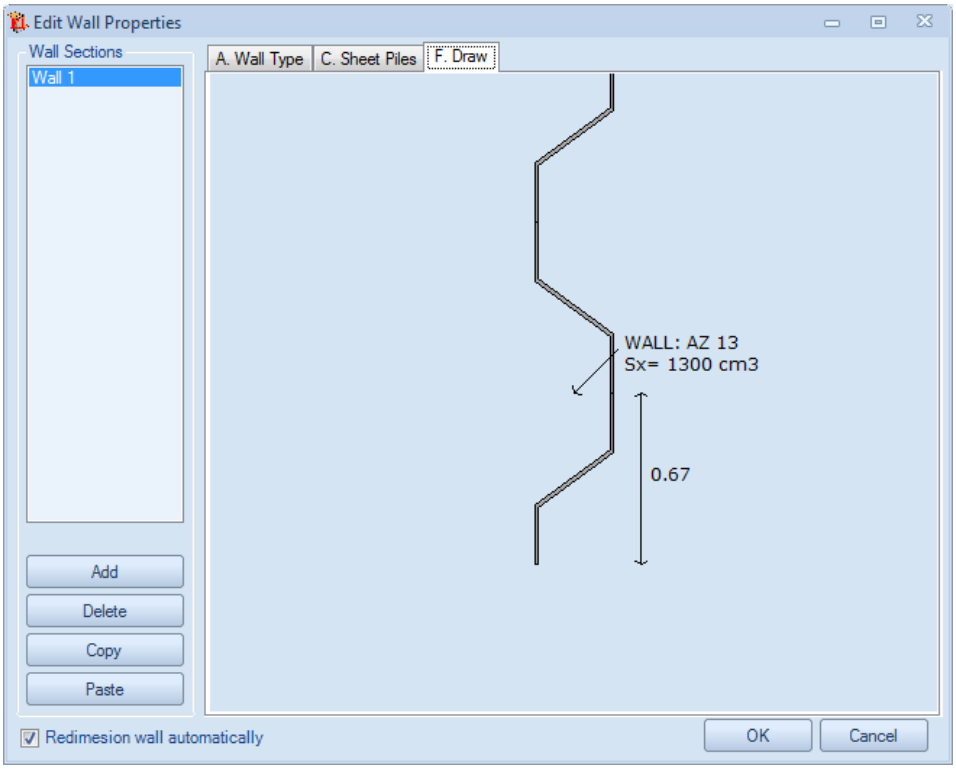


Figure 3.7.15: Sheet pile wall plan.

3.7.4 Wall type: Secant - tangent piles

At the wall sections dialog, we can select to use secant pile or tangent pile walls. The user can choose to use a standard steel section for the piles from the area marked in Figure 3.7.16. DeepXcav includes all European and US standard sections.

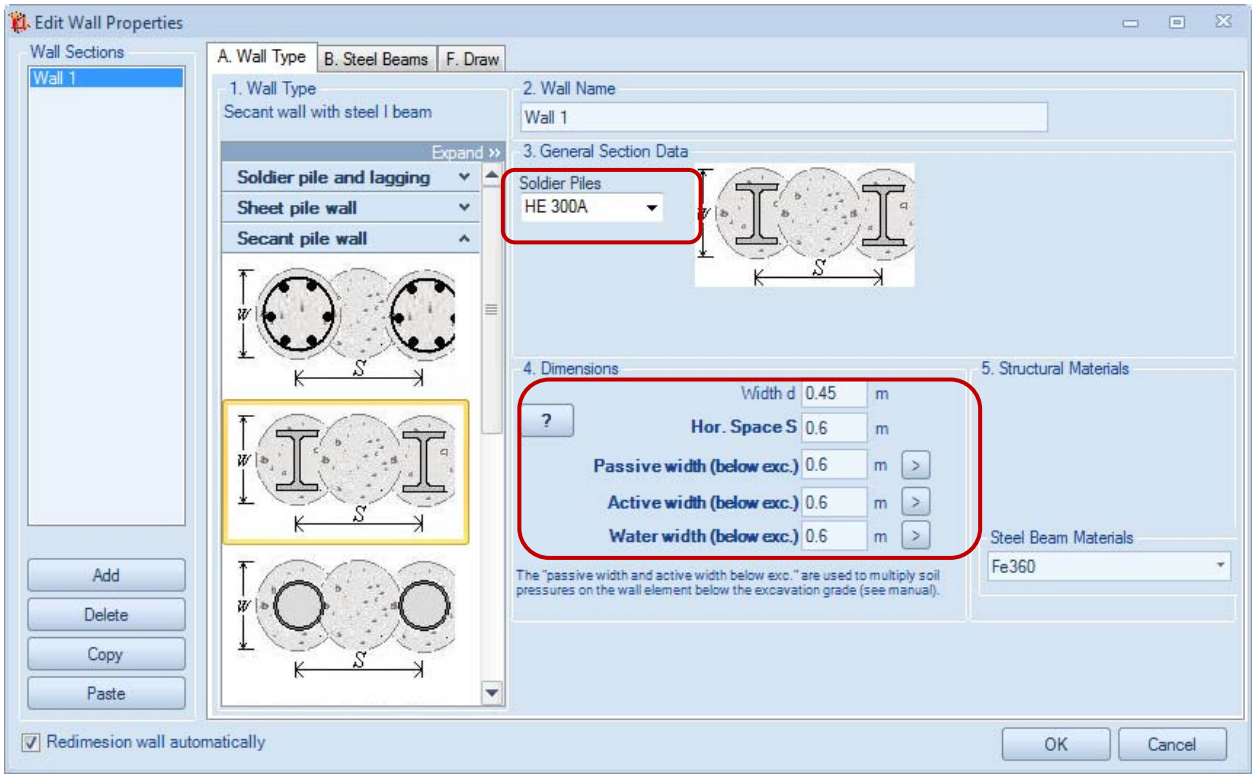


Figure 3.7.16: General steel section and dimensions for secant pile walls.

In B. steel beams tab (Figure 3.7.17), the user can modify the steel section properties. These properties are presented and described in the table below. The section can either be a normal T-section (Figures 3.7.18 and 3.7.19), a channel section with one or two channels (Figures 3.7.20 and 3.7.21) or a Pipe section, filled or not with concrete (Figures 3.7.22 and 3.7.23).

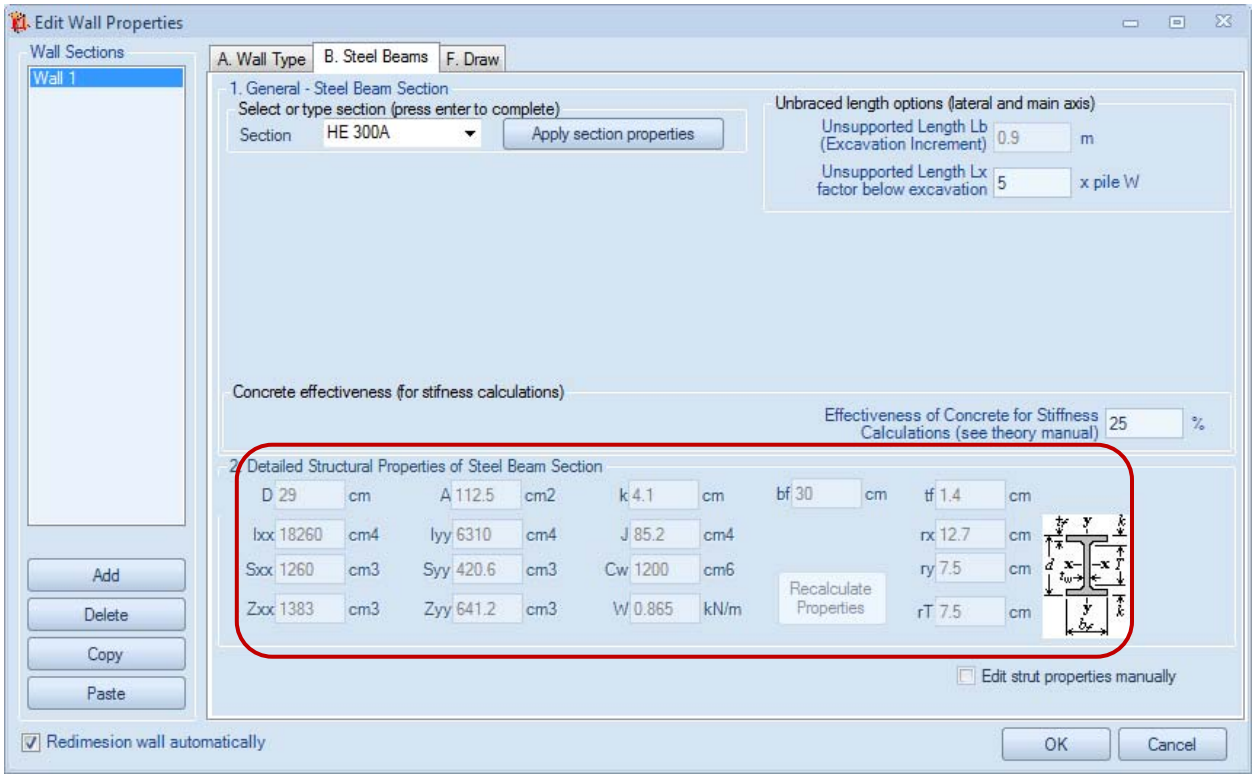


Figure 3.7.17: Steel section properties.

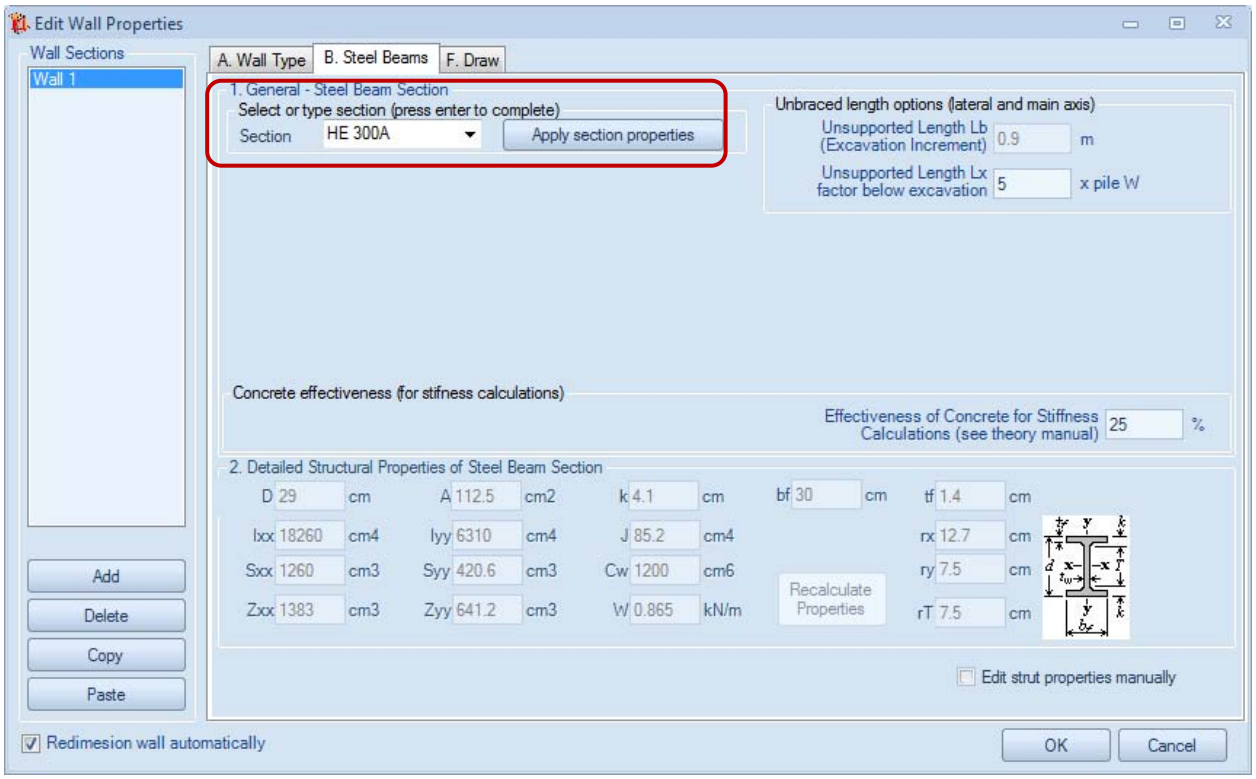


Figure 3.7.18: H-sections.

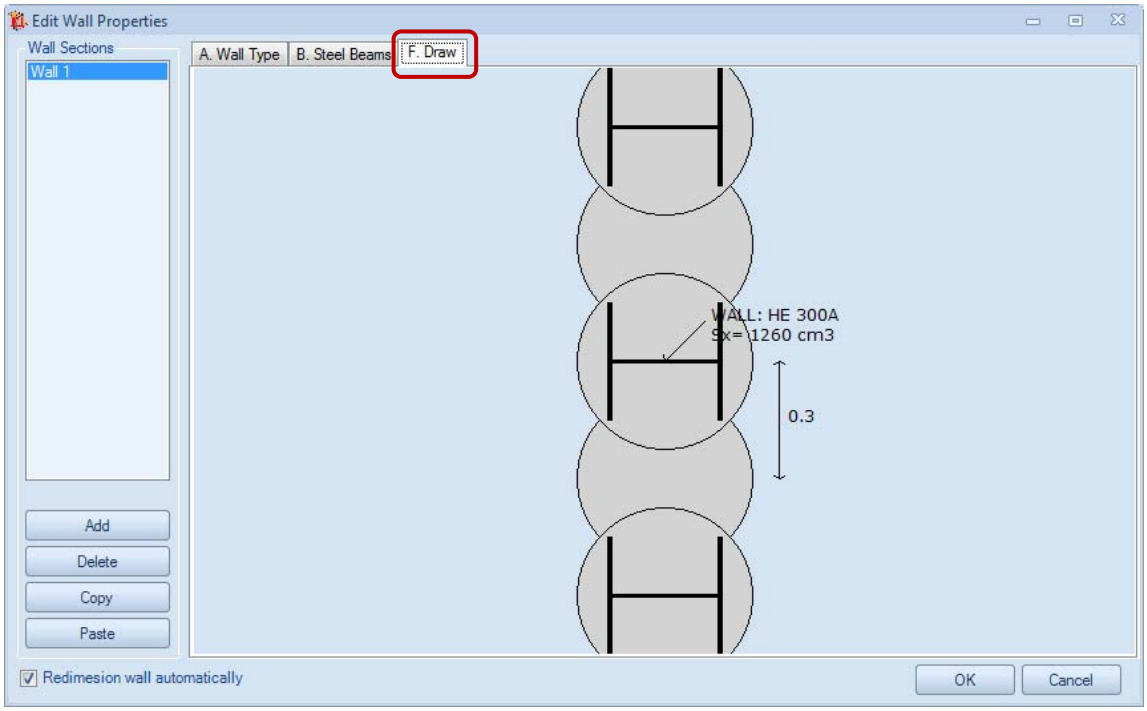


Figure 3.7.19: H-section secant pile wall plan.

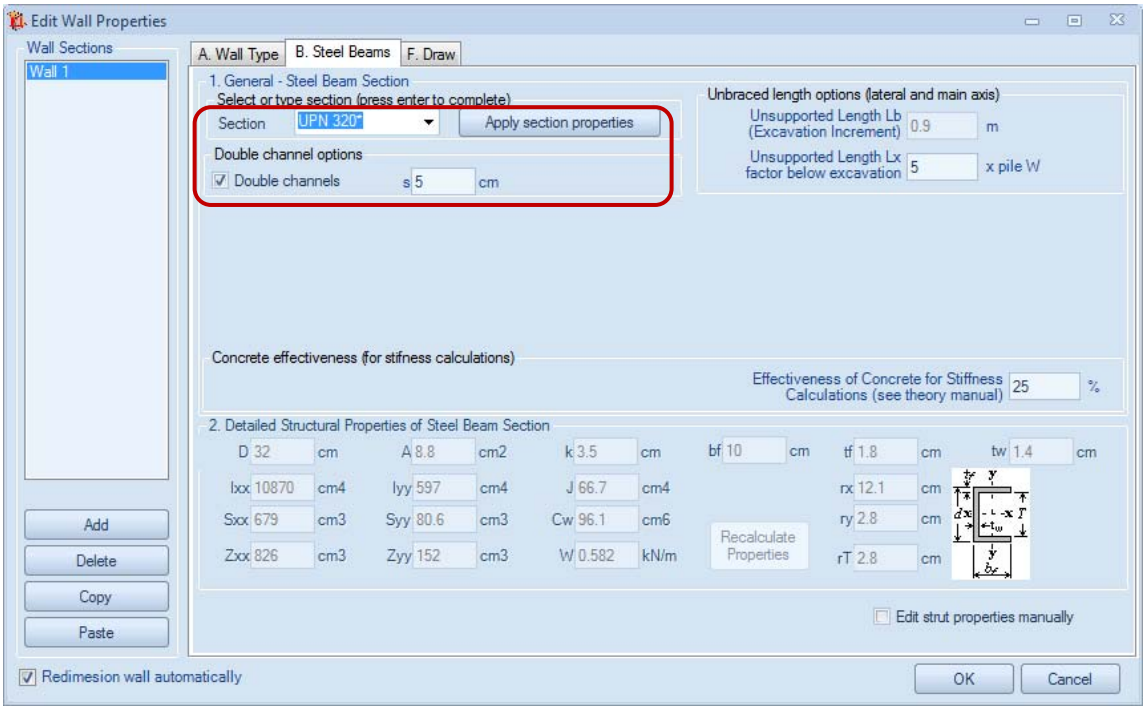


Figure 3.7.20: Double channel sections.

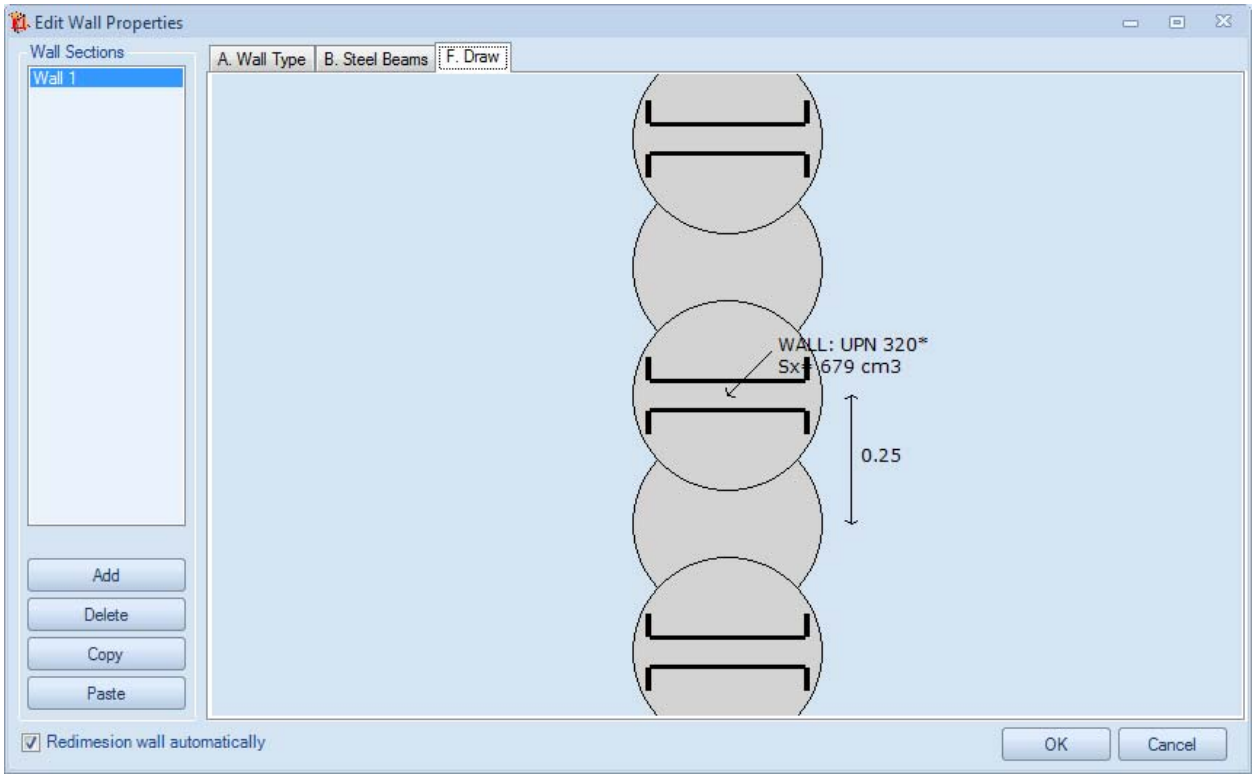


Figure 3.7.21: Double channel section secant pile wall plan.

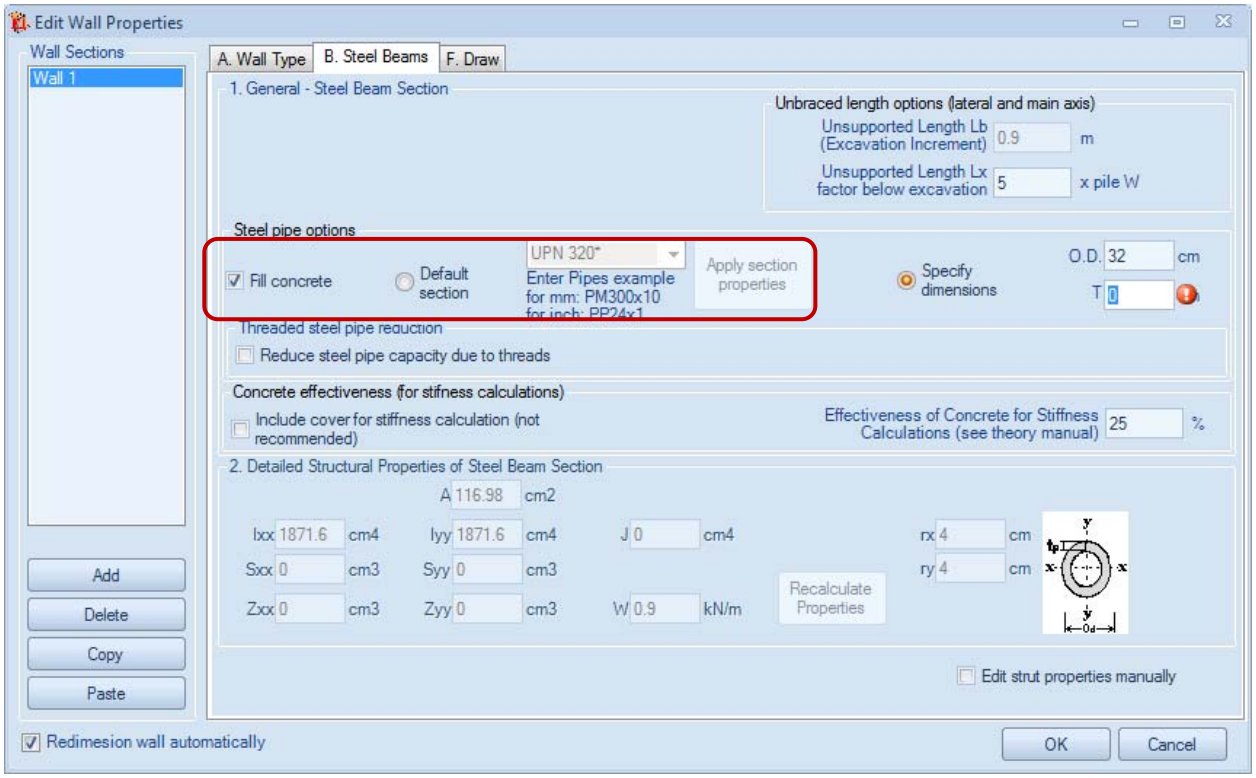


Figure 3.7.22: Pipe sections filled with concrete.

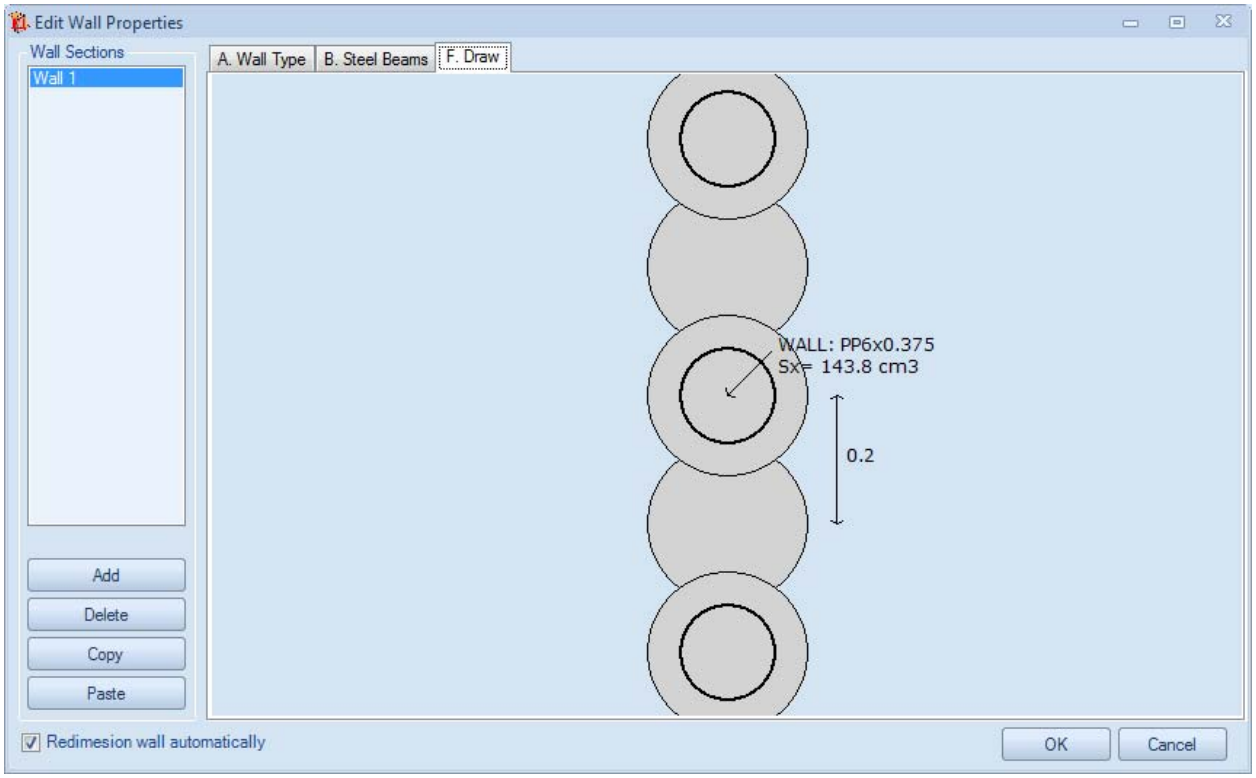


Figure 3.7.23: Pipe section filled with concrete secant pile wall plan.

3.7.5 Wall type: Diaphragm walls (slurry walls US)

At the wall type, we can select to use diaphragm walls. We can modify the dimensions of the wall in the marked area of Figure 3.7.24. In D. Concrete –Rebar tab we can change the concrete and rebar steel properties and define the wall's reinforcement (Figure 3.7.25.a). When the option to use more than one reinforcement sections is selected, the multiple reinforcement options tab appears (Figure 3.7.25.b). The properties included in this tab are presented and described in the table below. Figure 3.7.26 shows the plan of a sheet pile wall as presented in F. Draw tab.

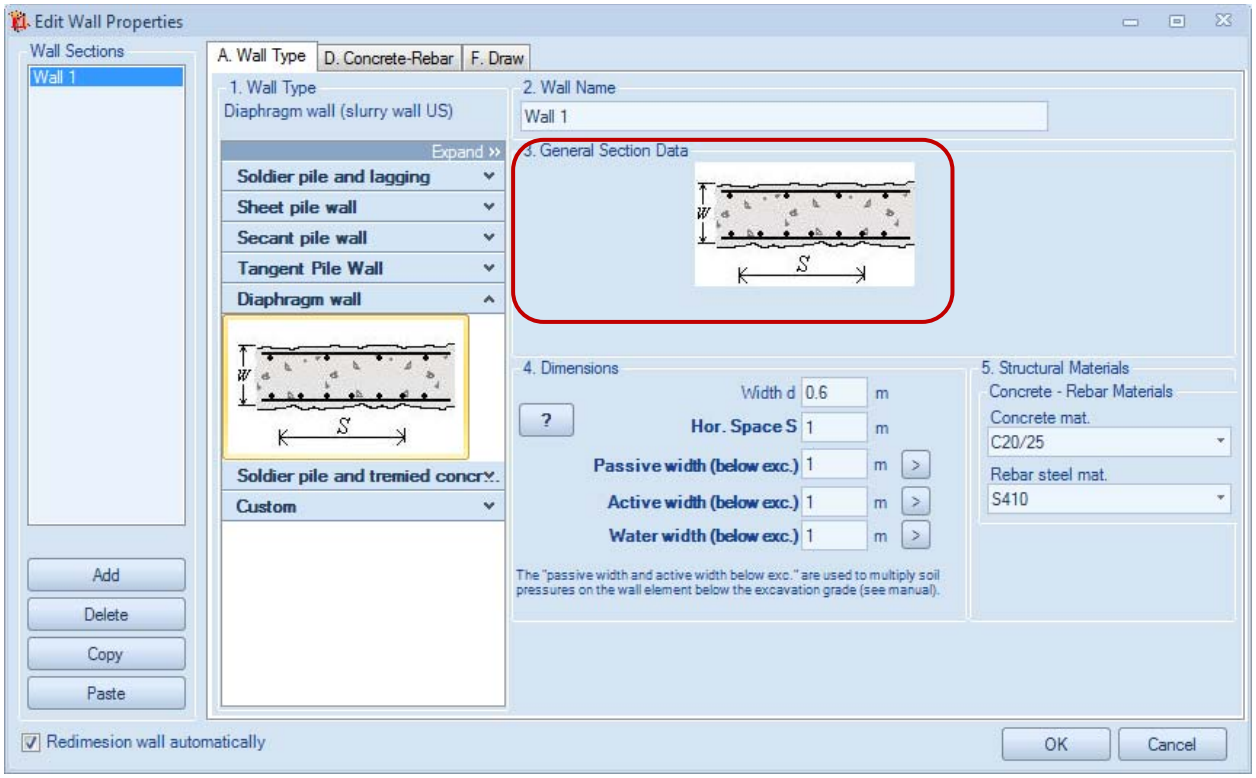


Figure 3.7.24: Dimensions for diaphragm walls.

Symbol	Description
D	Wall thickness (height)
A	Area
B	Wall bare length
Bw	Wall length
Ixx	Strong axis moment of inertia
N	Number of reinforcement bars
Bar#	Rotational moment of inertia
As top	Top reinforcement area
As bot	Top reinforcement area
Ctop	Top clearance to center of bar
Cbot	Bottom clearance to center of bar
sV	Vertical spacing of shear reinforcement
sH	Horizontal spacing of shear reinforcement
Use more than one reinforcement sections	Option that turns on the use of multiple reinforcement for the wall
Treat wall as slab	Option that is used only for diaphragm walls and allows them to be treated as slabs for the shear capacity calculations

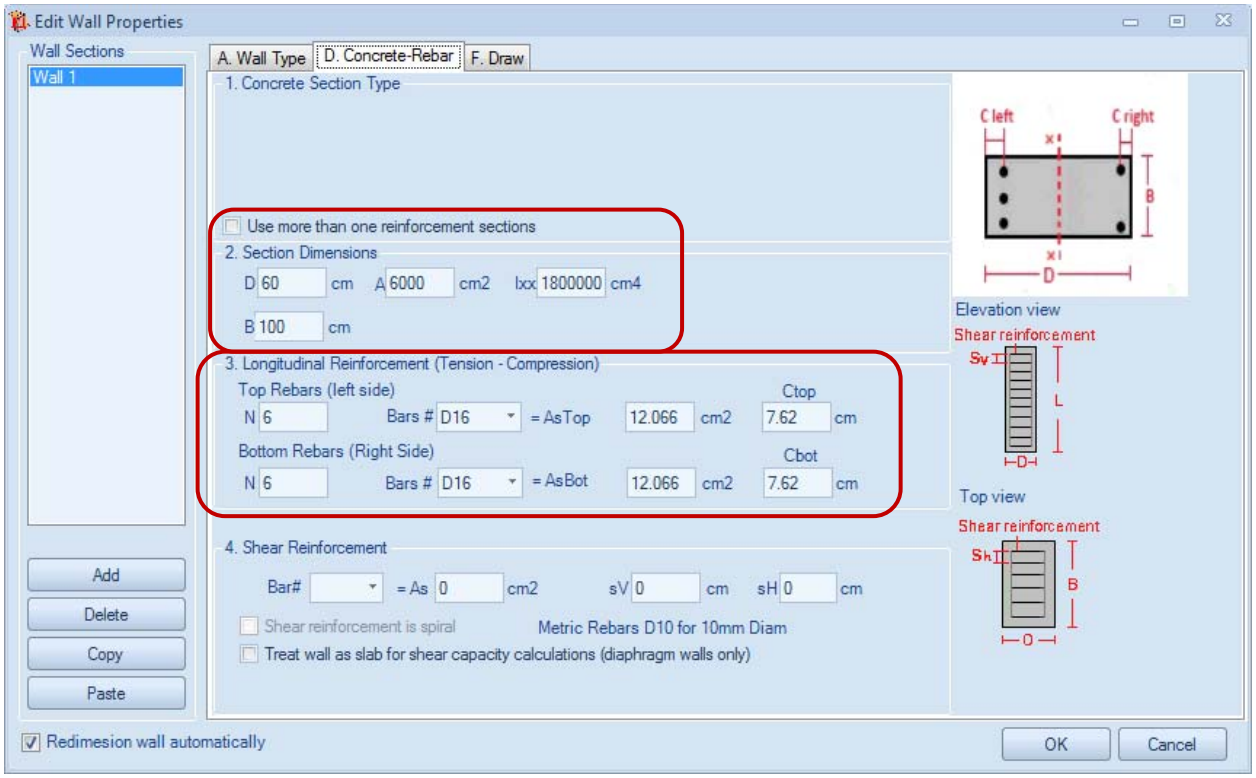


Figure 3.7.25.a: Concrete and rebar properties.

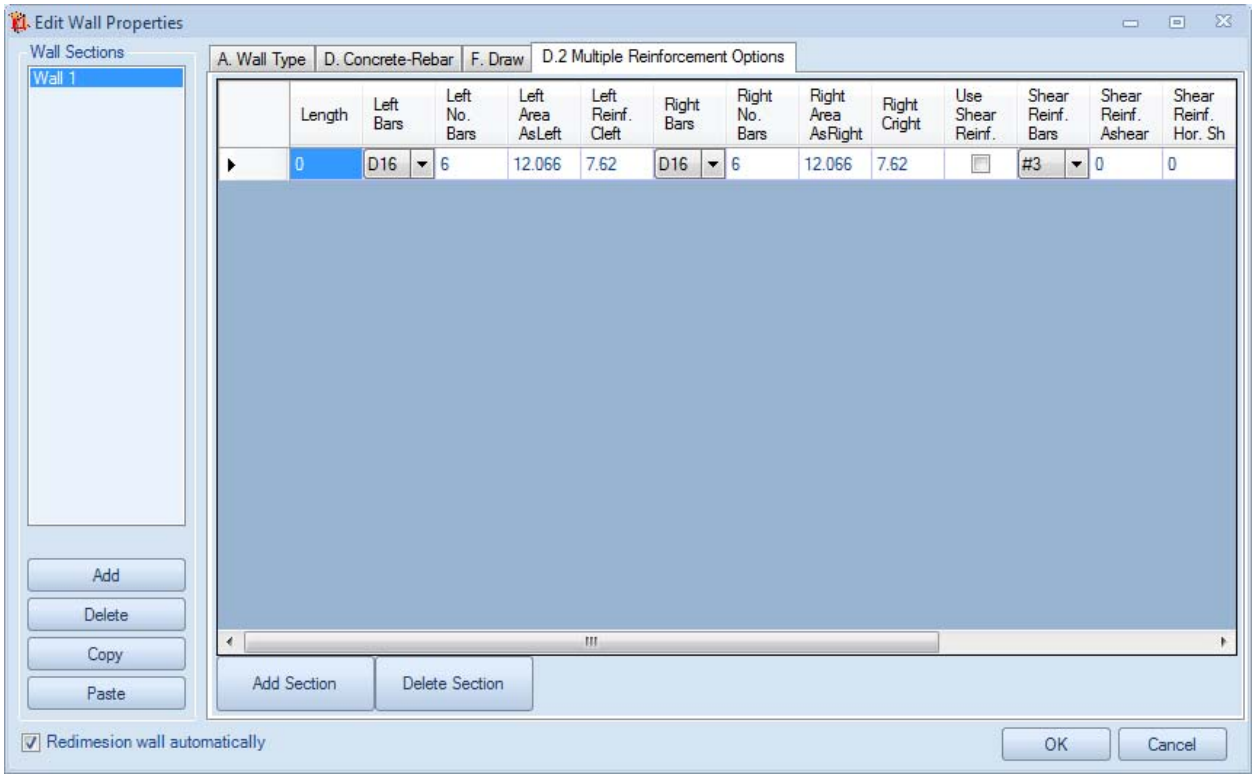


Figure 3.7.25.b: The multiple reinforcement options tab.

The following table presents the properties that are available in the multiple reinforcement options tab:

Length of the reinforcement (starting from top)
Left bar size
Number of left bars
Area of left bars
Clearance for the left bars
Right bar size
Number of right bars
Area of right bars
Clearance for the right bars
Option to use shear reinforcement
Dimension of shear reinforcement bars
Area of shear reinforcement
Horizontal spacing of shear reinforcement
Vertical spacing of shear reinforcement
Option to use and edit multiple reinforcement layers

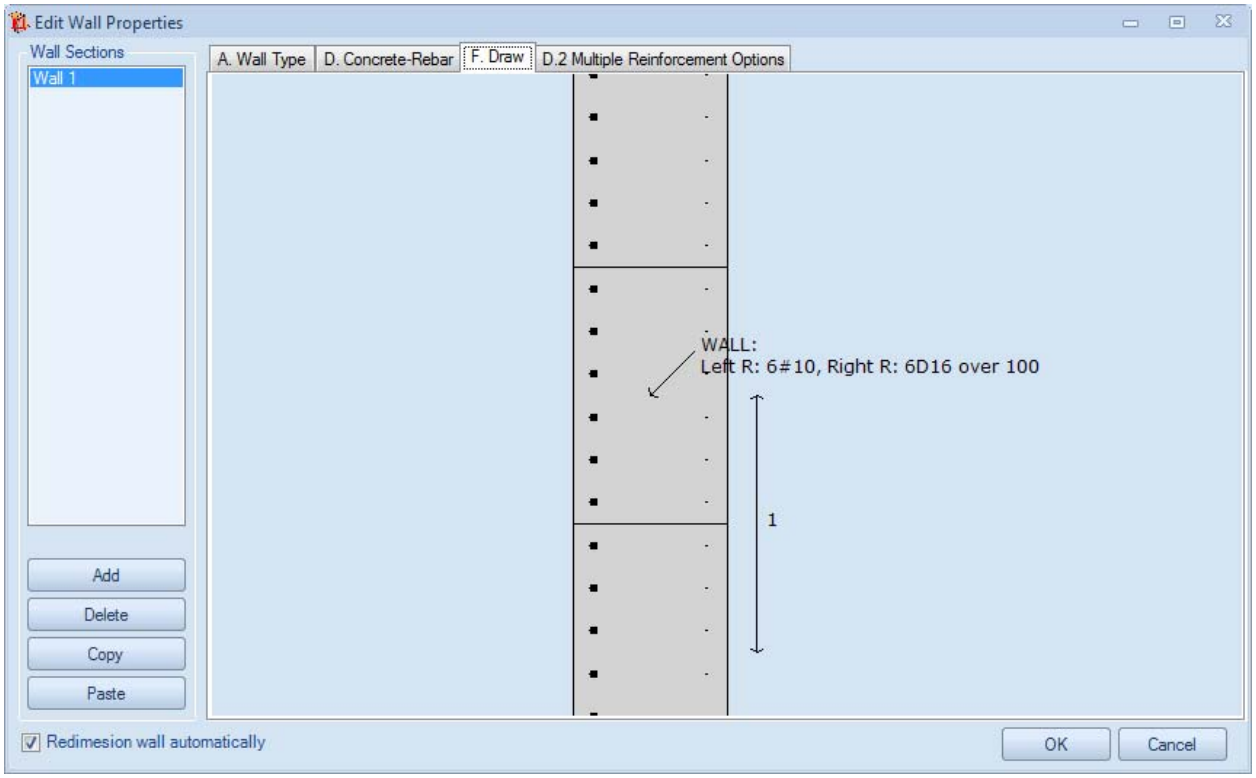


Figure 3.7.26: Diaphragm wall plan.

3.7.6 Wall type: Soldier pile and tremied concrete walls

At the wall sections dialog, the user can choose to use soldier pile and tremied concrete walls. The user can choose to use a standard steel section from the area marked in Figure 3.7.27. DeepXcav includes all European and US standard sections.

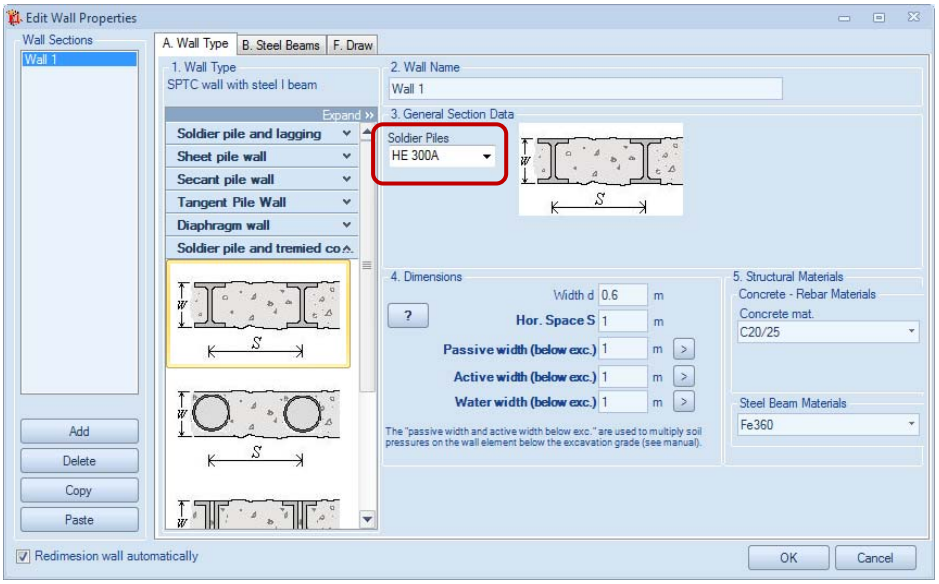


Figure 3.7.27: General section and dimensions for Soldier pile and tremied concrete walls.

The steel beam properties and options are the same as described in paragraph 3.7.1. Figure 3.7.28 shows the plan of a sheet pile wall as presented in F. Draw tab.

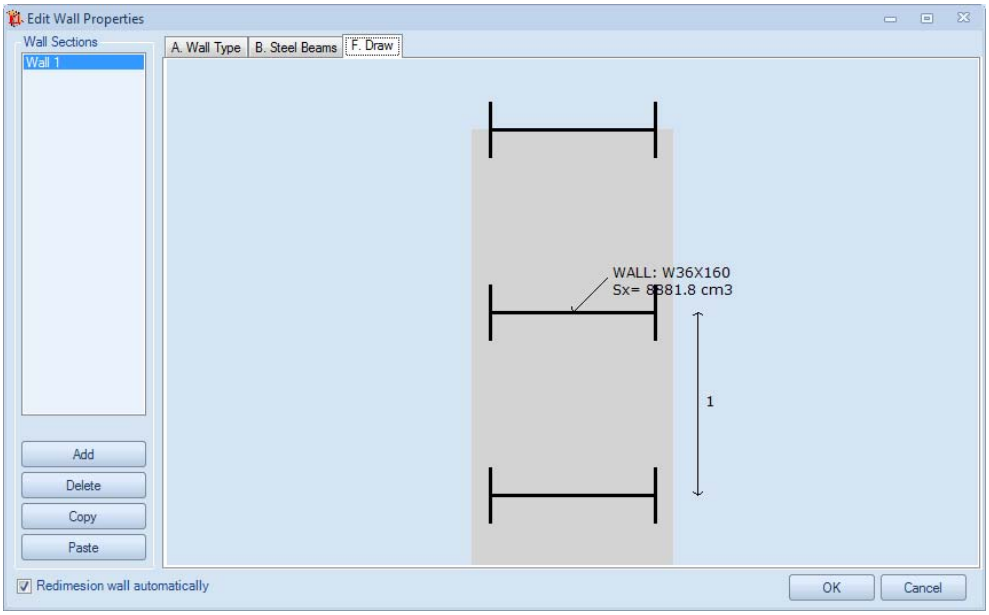


Figure 3.7.28: Soldier pile and tremied concrete wall plan.

3.7.7 Wall type: Custom

By using this wall type the user can define directly the wall moment of inertia, the wall thickness and the top and bottom wall releases without using any of the available wall types. Figures 3.7.29 and 3.7.30 show the custom wall properties and the custom wall plan respectively.

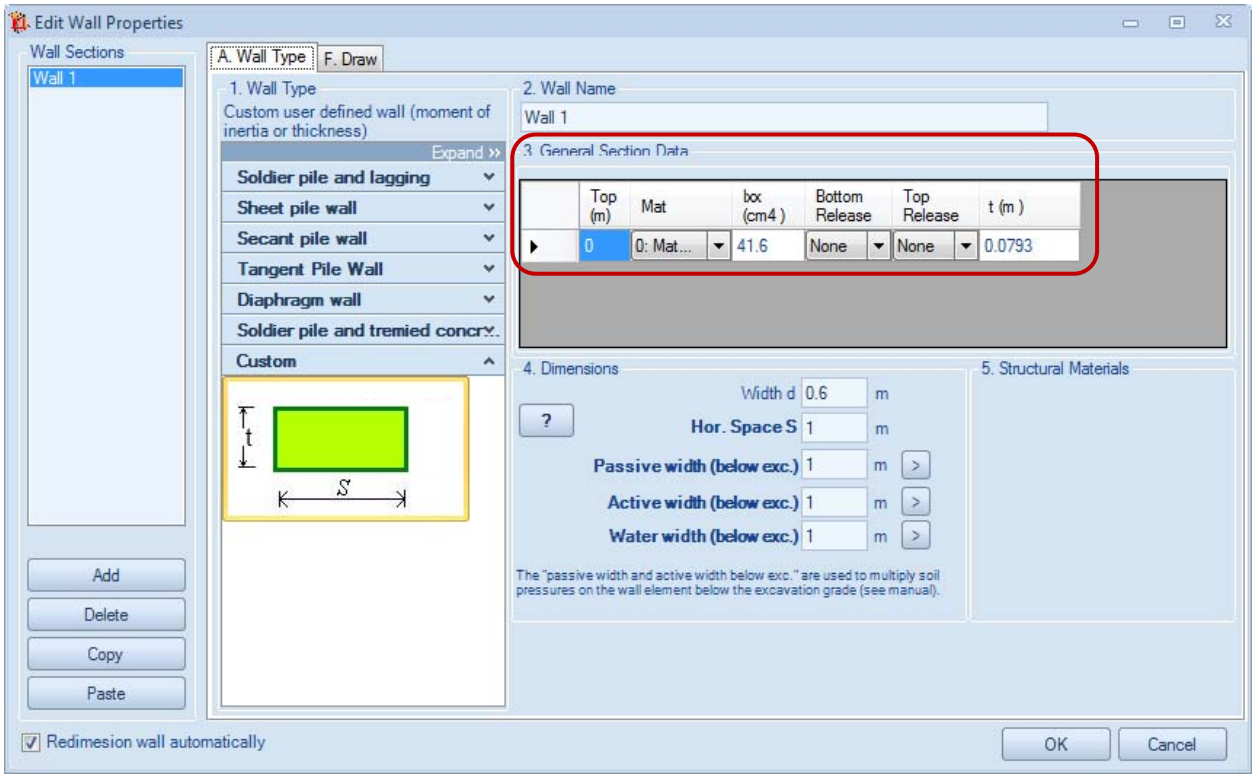


Figure 3.7.29: Custom wall properties.

Top	Top elevation of section
Mat	Section material
Ixx	Moment of inertia
Bottom release	Bottom or the section releases
Top release	Top of the section releases
t	Section thickness

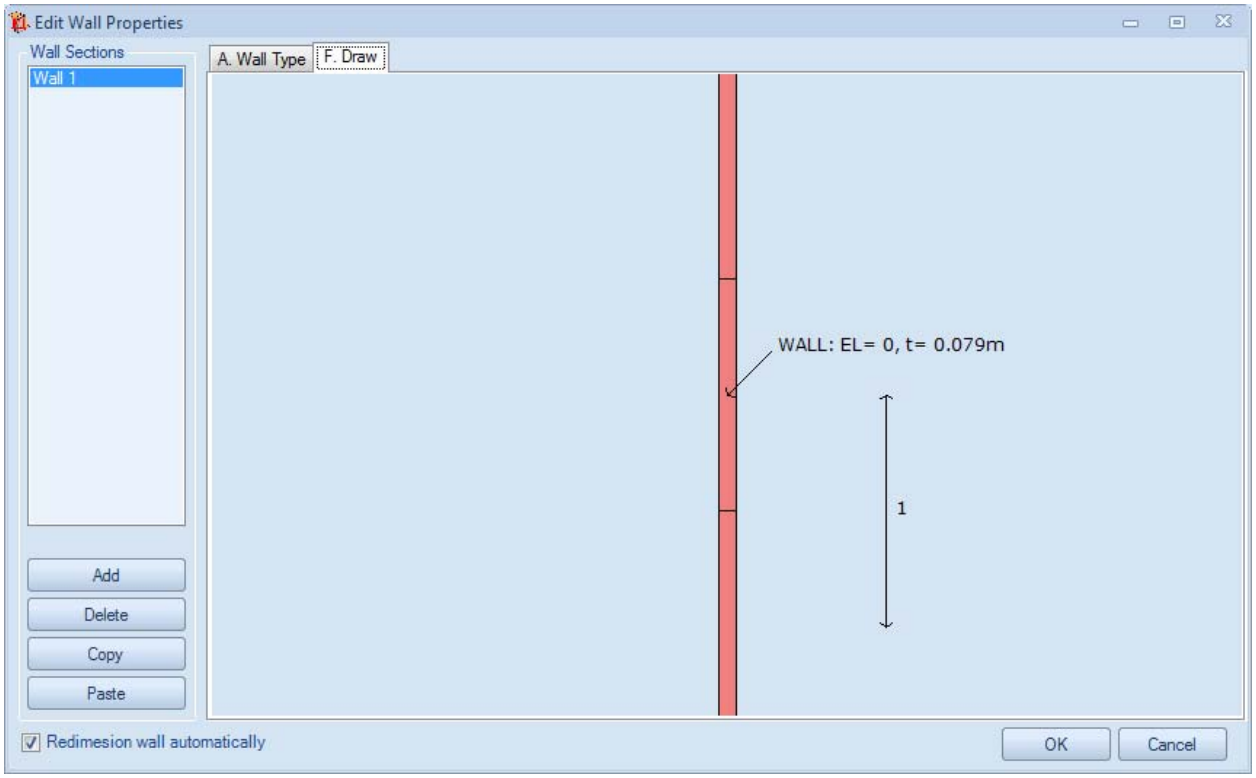


Figure 3.7.30: Custom wall plan.

3.8 DATA ENTRY

3.8.1 Data entry: Tiebacks

By choosing to add a tieback to the model, the Edit support dialog appears (Figure 3.8.1). In this area we can define the precise support coordinates, the free and the fixed length. In addition, the user can define the support's installation angle, the horizontal spacing of the tiebacks and the section used in each support. This section can be edited by clicking on the edit button.

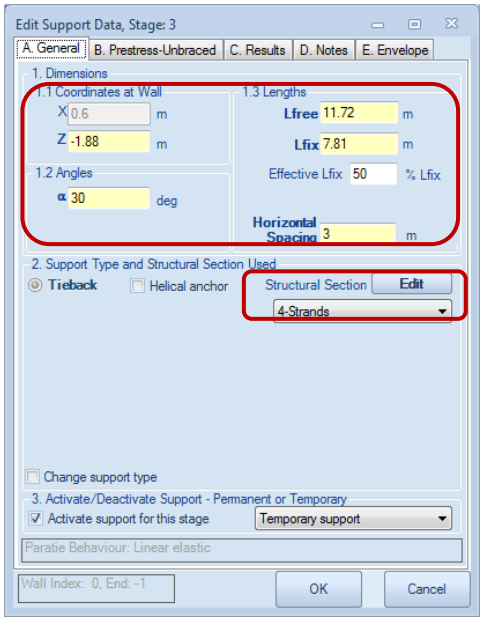


Figure 3.8.1: Edit support data dialog.

Here the user can define the following properties:

Symbol	Description
X	The X-coordinate of the support
Z	The Z-coordinate of the support
Lfree	Free length of the tieback
Lfix	Fixed length of the tieback
Effective Lfix	Effective fixed length, defined as a percentage of Lfix
a	Tieback installation angle
Horizontal Spacing	Horizontal spacing of installed tiebacks
Structural section	Define structural section of the tieback used
Helical anchor	Option to use helical anchor sections for the tieback
Change support type	Option to use a different type of support
Activate/Deactivate	Choose to activate/deactivate the support for the current stage
Temporary support	Define if the support is temporary or permanent

3.8.2 Data entry: Tieback sections

By clicking the Edit Section button of the support dialog, the Tieback – Anchors dialog appears (Figure 3.8.2).

In this dialog we can define the strand material and diameter, the grout options and a geotechnical factor of safety. In most cases, tiebacks are pressure grouted in order to achieve greater capacities. We can capture such effects by including a densification multiplier for the frictional and cohesional components of the shear resistance. These factors require careful consideration and can be used to obtain a preliminary estimate of geotechnical tieback capacity (Figure 3.8.3). In the overwhelming majority of projects, all tiebacks have to be tested. For the cohesional component you can select an adhesion factor behavior (multiplier for the cohesion). The frictional component of the tieback capacity is calculated from the average effective confining stress at the tieback (at-rest on sides + vertical on top & bottom). Bearing capacity is ignored. The tieback section can be selected from three main types:

A) Strands or user bars

B) Solid bar

C) User area

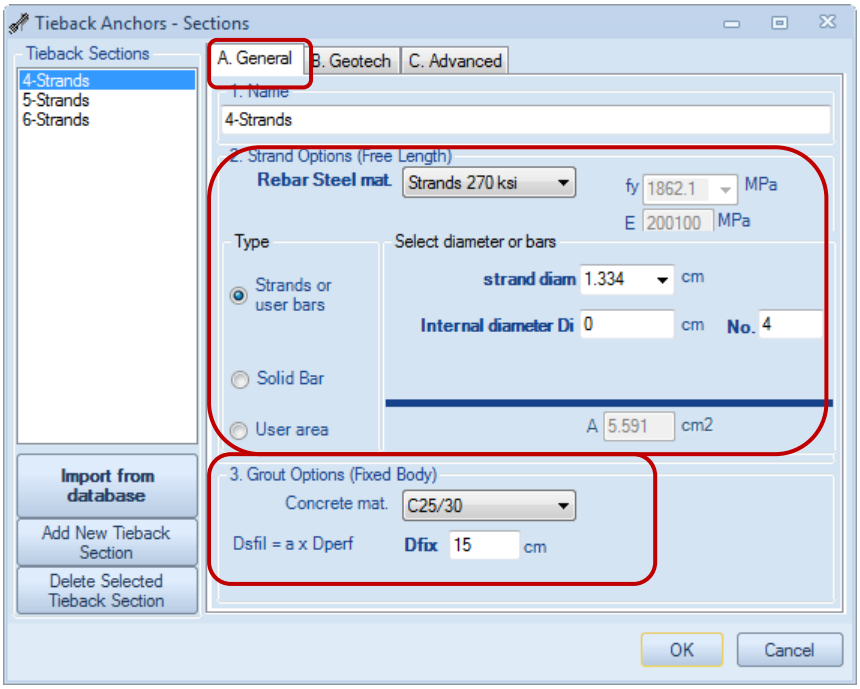


Figure 3.8.2: Tieback - anchor sections dialog.

Here the user can define the following properties:

Symbol	Description
Name	Define the section name
Rebar steel mat	Define the rebar steel material
fy	The yield strength for the steel
E	Modulus of elasticity
Strand diam	Diameter for strands (if strands option is selected)
Di	Internal strand diameter (if strands option is selected)
No	Number of strands used (if strands option is selected)
Bar #	The bar diameter (if solid bars option is selected)
No	Number of bars used (if solid bars option is selected)
A	Reinforcement area (if user area option is selected)
Concrete mat	Define the concrete material used as grout
Dfix	Fixed (grouted) diameter. This diameter is usually a function of the performed diameter (or drill diameter)

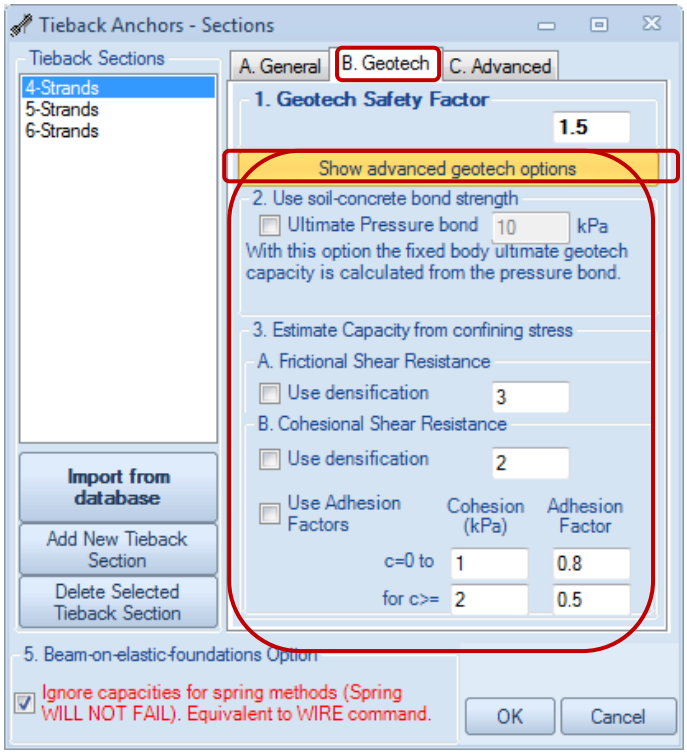


Figure 3.8.3: Advanced geotechnical options.

The geotechnical safety factor in this dialog is used when the custom Geo FS option is not selected in the mail program.

Here the user can define the following properties:

Symbol	Description
Ultimate Pressure bond	Define ultimate pressure bond (if the option is selected), so that the ultimate geotechnical capacity to be calculated from the pressure bond
-	Use and define densification of frictional shear resistance (if option is selected)
-	Use and define densification of cohesional shear resistance (if option is selected)
Adhesion factor	Define adhesion factors (if the option is selected)

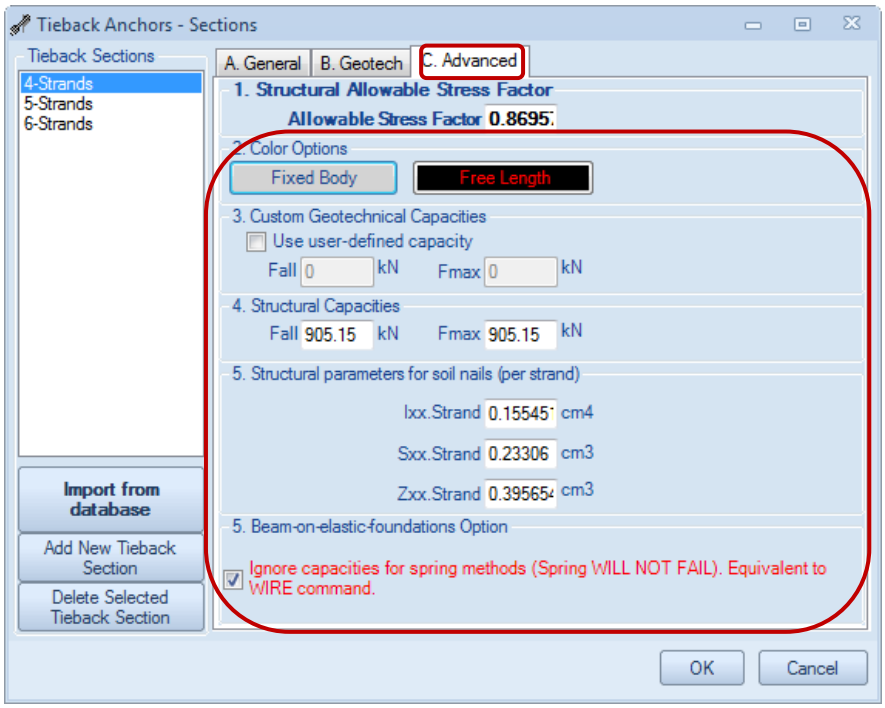


Figure 3.8.4: Advanced options.

Here the user can define the following properties:

Symbol	Description
-	Define the structural allowable stress factor for steel
-	Define the color of the free length and fixed body
Fall	Define the allowable geotechnical capacity (if the use user defined capacity option is selected)
Fmax	Define the maximum geotechnical capacity (if the use user defined capacity option is selected)
Fall	Define the allowable structural capacity
Fmax	Define the maximum structural capacity
Ixx.strand	Moment of inertia of the strand
Sxx.strand	Elastic section modulus for one strand (used in soil nails)
Zxx.strand	Plastic section modulus for one strand (used in soil nails)
-	Option to ignore capacities for spring methods (will create a npn-yielding element, recommended option)

After defining the tieback section properties, the user should define the tiebacks prestress in the edit support dialog (Figure 3.8.5). If prestress is applied, it must be applied ONLY in the support installation stage.

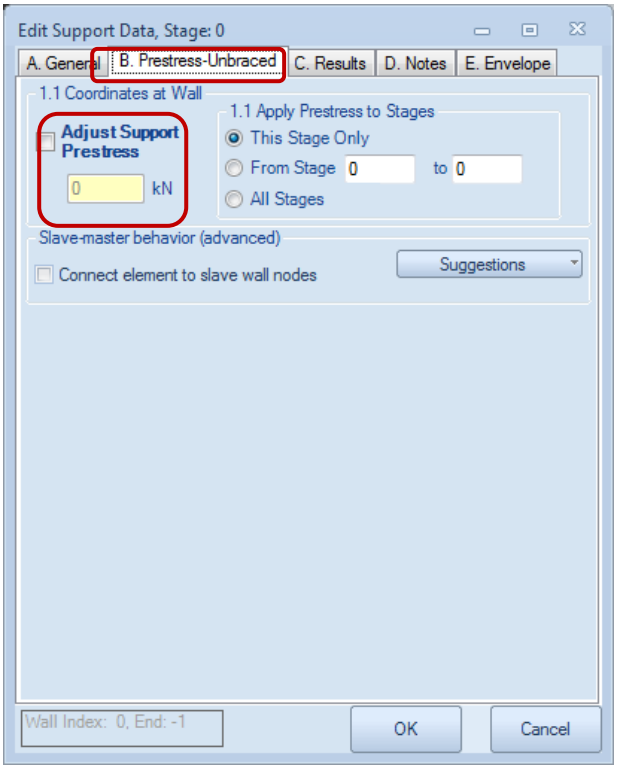


Figure 3.8.5: Tieback prestress.

At the Results tab the user can see the results of the support, after the model is calculated.

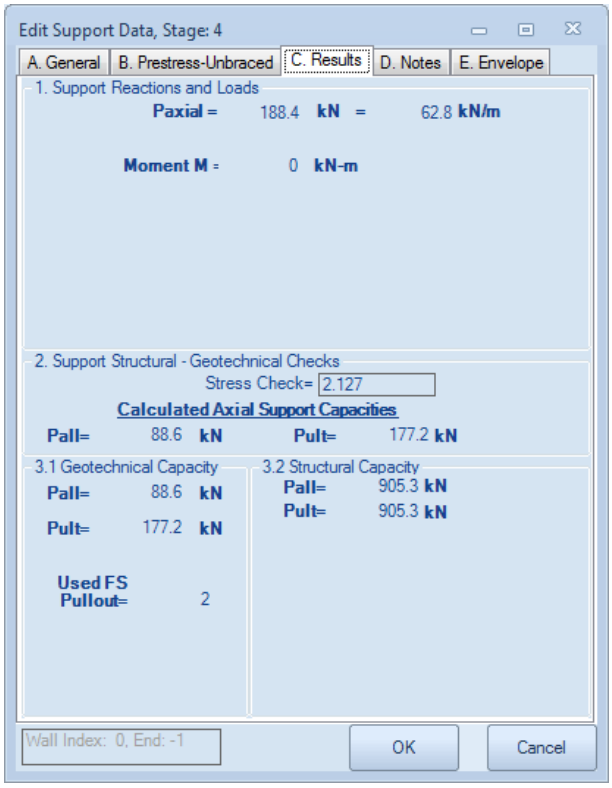


Figure 3.8.6: Tieback results.

The following results are available:

Symbol	Description
Paxial	Axial load of the tieback support
M	Moment
Pall	Allowable structural axial capacity
Pult	Ultimate structural axial capacity
Pall	Allowable structural axial capacity
Pult	Ultimate structural axial capacity
Used FS	Overall applied safety factor for geotechnical design capacity in terms of the ultimate capacity y

3.8.3 Data entry: Helical anchor sections

From the Helical Anchors dialog (Figure 3.8.7) the user can select an anchor from the existing database of anchors or add some sections to the database. These anchors can use in the project. The General tab contains basic structural information. The Geotechnical capacity tab contains options for calculating the geotechnical tieback capacity. In the Advanced tab the user can specify safety factors as well as customize the allowable and ultimate geotechnical and structural capacities. The properties included in this dialog are presented and described in the table below.

Symbol/Option	Description
Fy	Tensile Yield Strength of anchor
Fu	Tensile Ultimate Strength of anchor
Diameter	Anchor diameter
Thickness	Anchor thickness
Ixx	Moment of inertia
Sxx	Elastic section modulus
Zxx	Plastic section modulus
Telastic	Torsional elastic pipe capacity
Tplastic	Torsional plastic pipe capacity
E	Modulus of elasticity
Apipe	Area of the pipe of the anchor
Qyield	Tensile yield shaft capacity
Qultimate	Tensile ultimate shaft capacity
Helix diameter	The diameter of the helical plate
Helix spacing	The spacing between the helical plates
Helix thickness	The thickness of the helical plate
Effective helix area	The effective area of the helical plate
Helix pitch	The helical plate inclination
Qhelix	Ultimate tension capacity for one helical plate
FS	Factor of safety
σ' vmax	Limiting vertical stress

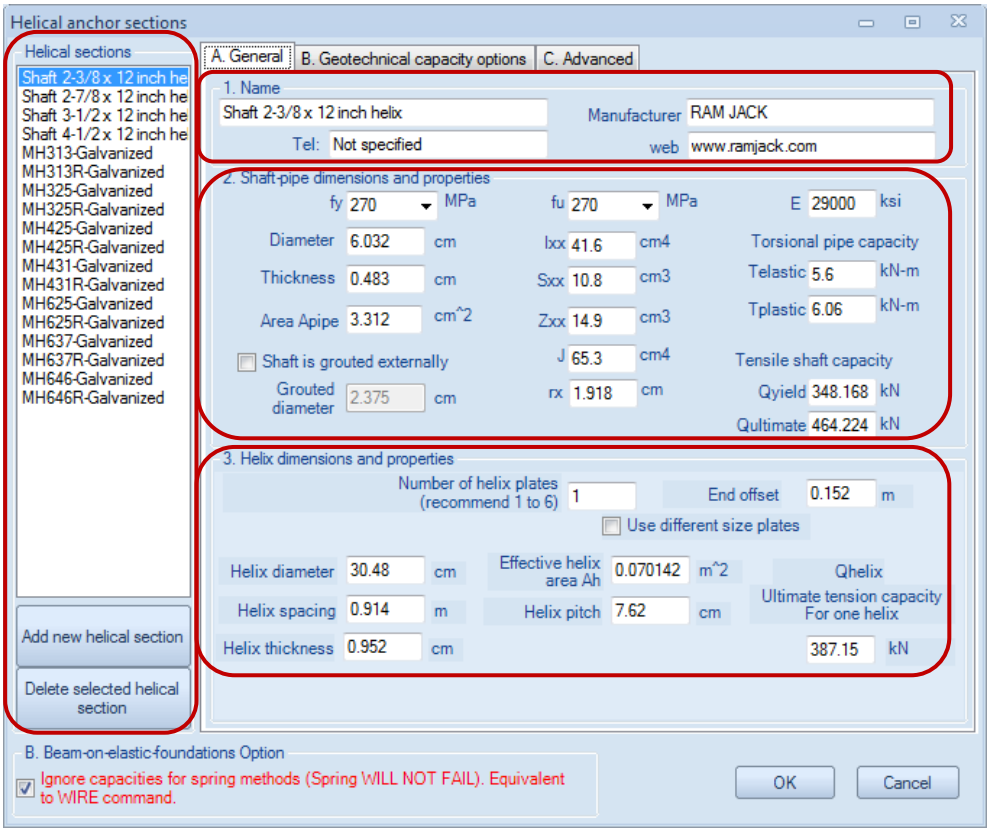


Figure 3.8.7: Helical anchor sections dialog.

3.9 Data entry: Strut sections

The strut section properties can be modified either by clicking on the Edit strut sections button

of the Properties tab, or by clicking on the Edit section button of the Edit support data dialog that opens when the user adds a raker or a strut to the model. Struts and rakers in DeepXcav can be either H-section steel beams, or pipes. DeepXcav includes all European and US standard sections. The user can choose a section from the program's database or edit the strut properties manually (Figure 3.9.1) The properties are presented and described in the following table.

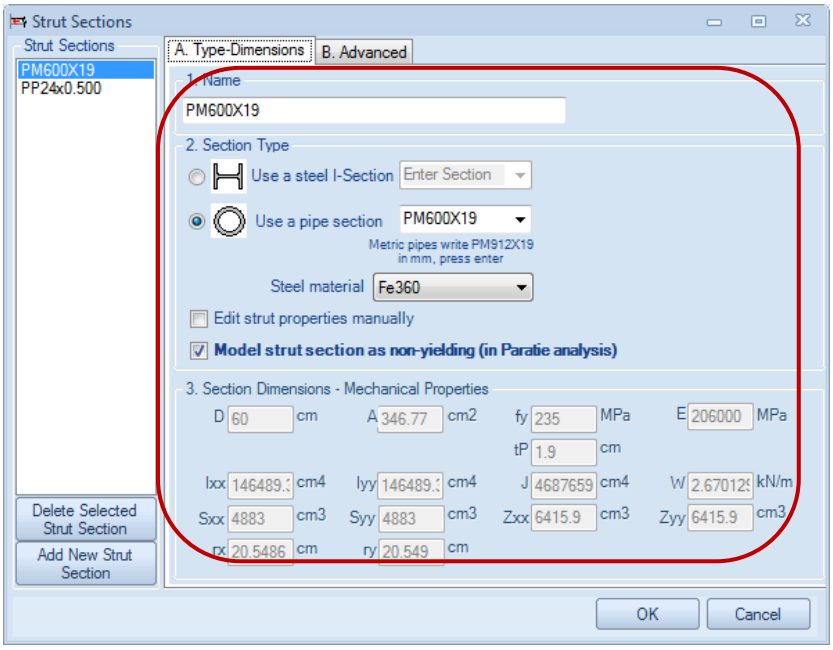


Figure 3.9.1: Strut sections dialog.

Symbol	Description
D	Depth or diameter
A	Area
fy	Yield strength
Ixx	Strong axis moment of inertia
Iyy	Weak axis moment of inertia
Sxx	Strong axis elastic section modulus
Syy	Weak axis elastic section modulus
rx	Strong axis radius of gyration
ry	Weak axis radius of gyration
Zxx	Strong axis plastic section modulus
Zyy	Weak axis plastic section modulus
tP	Thickness
J	Rotational moment of inertia
W	Wrapping constant

After defining the strut section properties, the user can define the strut's or raker's prestress in the edit support dialog (Figure 3.9.2). If prestress is applied, it must be applied ONLY in the support installation stage. For struts a negative prestress would indicate compression (recommended).

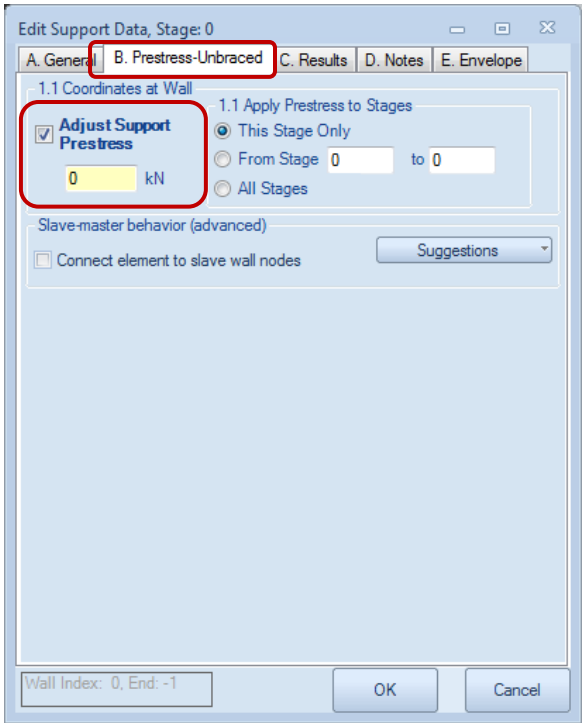


Figure 3.9.2: Strut prestress dialog.

3.10 Data entry: Slab sections and slab supports

The slab section properties can be modified either by clicking on the Edit slab sections button

of the Properties tab, or by clicking on the Edit section button of the Edit support data dialog that opens when the user adds a slab to the model. In A. Section Properties tab we can modify the concrete and rebar steel properties and define the wall's reinforcement (Figure 3.10.1). The properties included in this tab are described in the following table. Sometimes we may choose to define the slab's prestress in the edit support dialog (Figure 3.10.2). If prestress is applied, it must be applied ONLY in the support installation stage. In the same figure the area where we can define the slab's live load and the unbraced lengths is marked in red.

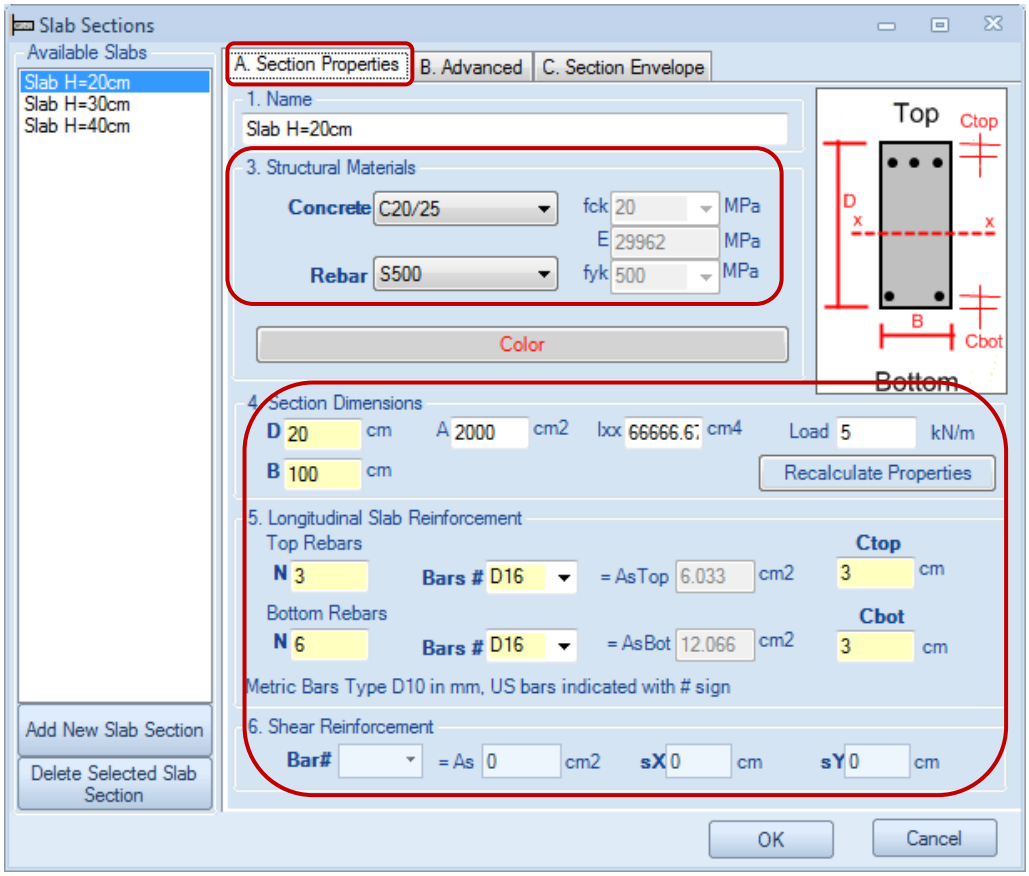


Figure 3.10.1: Slab Sections dialog.

Symbol	Description
D	Wall thickness
A	Area
B	Wall length
Bw	Wall length
Ixx	Strong axis moment of inertia
tf	Flange thickness
Bar#	Reinforcement area
As, top	Top reinforcement area
As, bot	Top reinforcement area
Ctop	Top clearance to center of top bar
Cbot	Bottom clearance to center of bottom bar
sX	Horizontal spacing of shear reinforcement
sY	Spacing of shear reinforcement in out of plane direction (in the model screen)

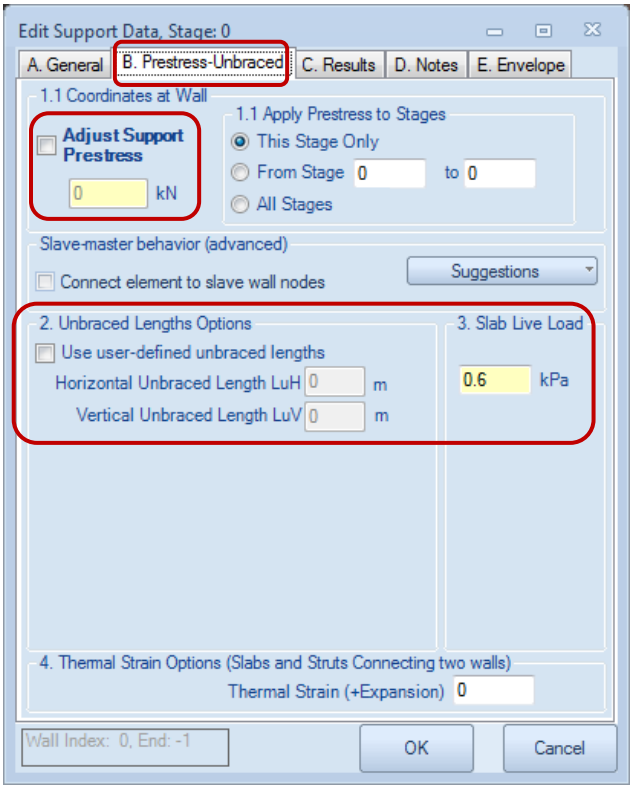


Figure 3.10.2: Slab prestress, live load and unbraced lengths.

At the C. Results tab, results for the slab are available when the model is calculated.

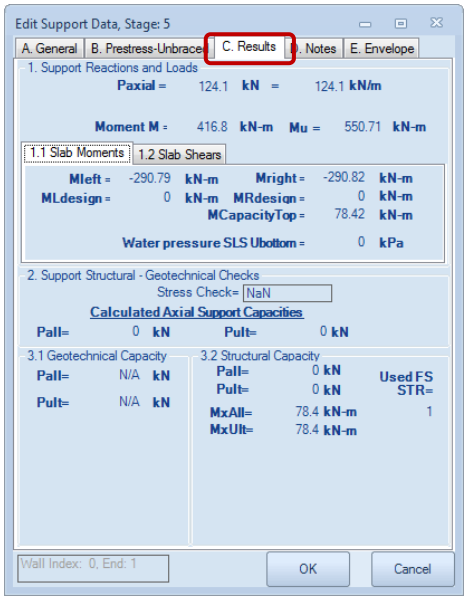


Figure 3.10.2: Slab results.

The following results are available in the results tab:

Symbol	Description
Paxial	Axial load on slab
M	Moment at center span
Pall	Allowable axial capacity
Pult	Ultimate axial capacity
Mleft	Left moment (>0 when slab is restrained)
MLdesign	Left side design moment (factored with safety factor)
Mright	Right side moment (>0 when slab is restrained)
MRdesign	Right side design moment (factored design moment)
Mcapacity top	Moment capacity of the slab for negative moments
Ubottom	Water pressure SLS (unfactored water pressure at bottom of slab when the slab is used to seal the excavation)
Vleft	Left side shear
VLdesign	Left side design shear
VLcap	Left side shear capacity
Vright	Right side shear
VRdesign	Right side design shear
VRcap	Right side shear capacity

In addition, we can see the geotechnical and the structural allowable and ultimate axial capacities of the slab.

3.11 Data entry: Fixed supports

We can choose to add a fixed support to the model by clicking on the button of the Loads/supports tab. A fixed support is a support that provides horizontal restraint. After selecting this tool, select any wall with your mouse where you want your support to be added.

3.12 Data entry: Springs

We can choose to add a spring support to the model by clicking on the button . After selecting this tool, select any wall with your mouse where you want your support to be added.

When the spring is added to the model, the Edit support data dialog appears. At B. Prestress – Unbraced tab we can define the axial and rotational spring stiffness (Figure 3.12.2).

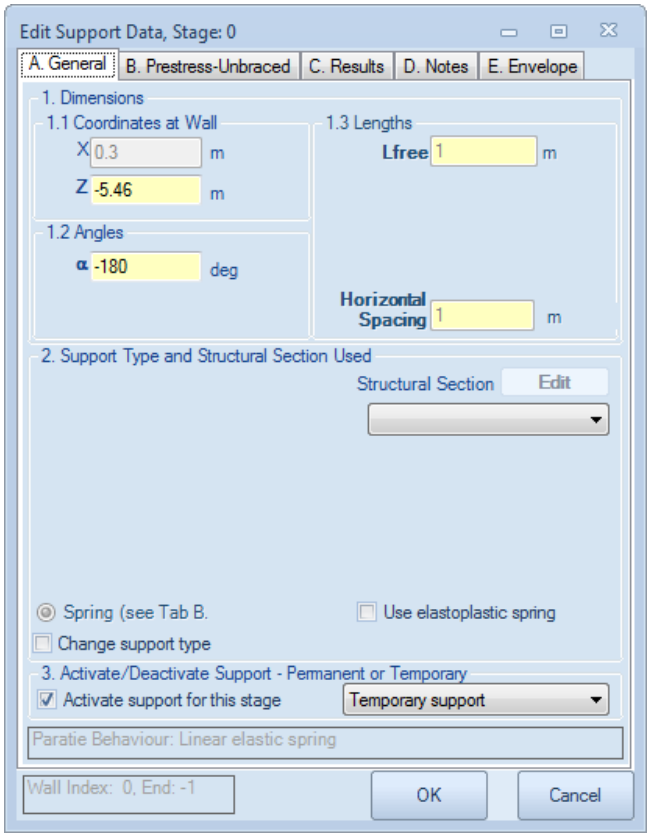


Figure 3.12.1: Spring support dialog.

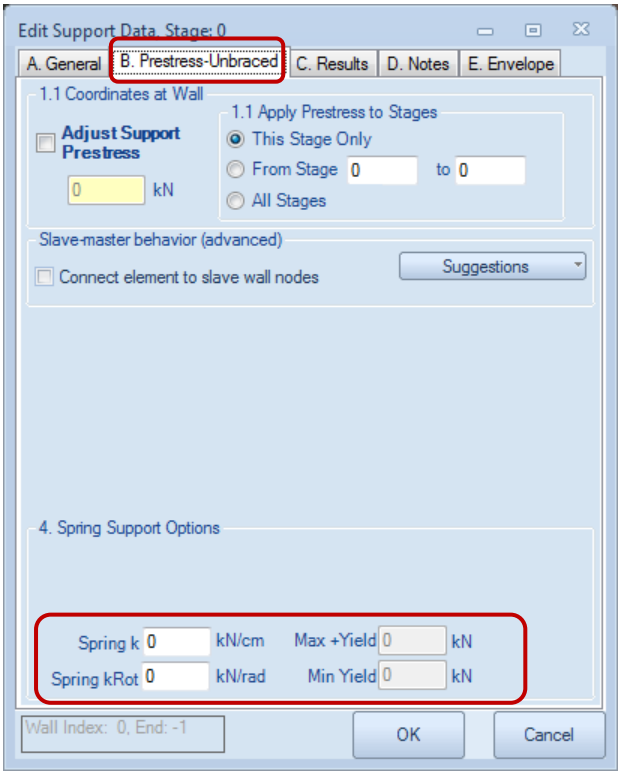


Figure 3.12.2: Define spring stiffness.

3.13 Data entry: Slope Stability

3.13.1 Slope Stability Analysis

The slope stability analysis can be performed using the Slope module of DeepXcav. The program can perform slope stability analysis of slope surfaces using the simplified Bishop method, the GLE (Morgenstern – Price) method or the Spencer (Force – Moment) method. To illustrate this a slope example will be presented. First, a slope or bench surface is created (right click on the model, set left bench surface). Then, the bench options dialog appears (Figure 3.13.1). Here we set the bench surface by specifying the bench angle or the Vertical – Horizontal step.

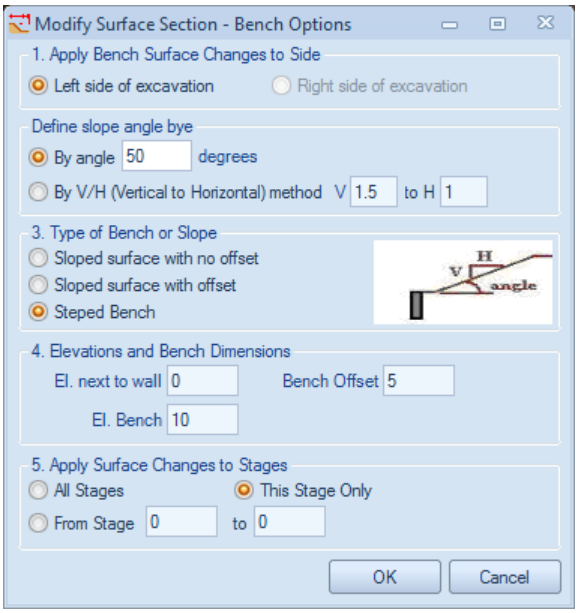


Figure 3.13.1: Set bench surface.

Next, we define the exact center or the rectangle containing the centers for the slope surfaces that to be analyzed (Figures 3.13.2 and 3.13.3).

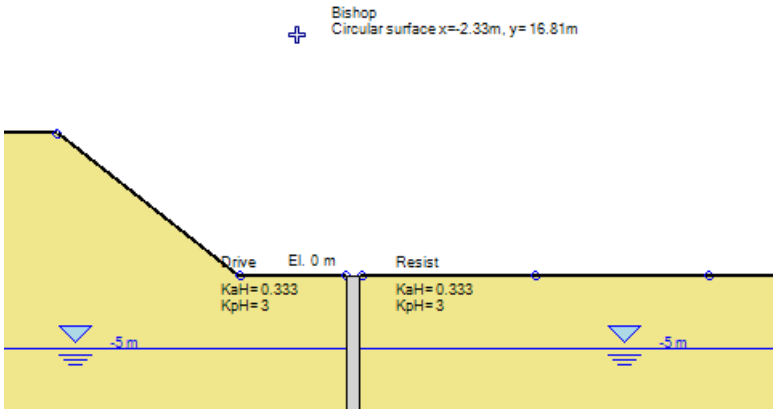


Figure 3.13.2: Slope circle center.

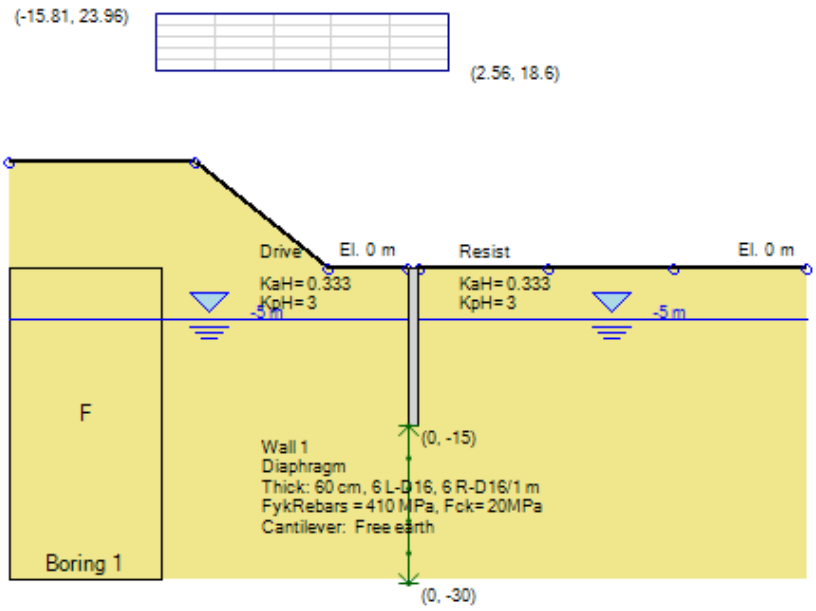


Figure 3.13.3: Rectangular center point search.

Next, the user should select the points where the sliding surface will pass. Using DeepXcav, the user can use the following options:

- Single search radius option (Figure 3.13.4): the user adds a single sliding surface to the model.

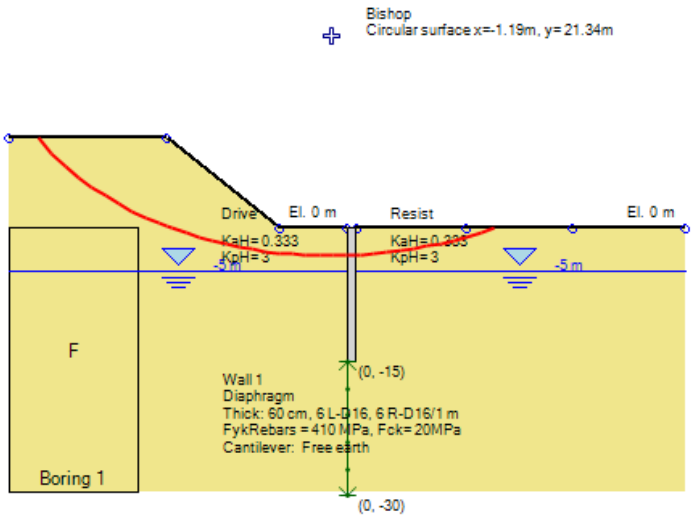


Figure 3.13.4: Single radius.

- Custom failure surface option (Figure 3.13.5). The user adds a custom sliding surface to the model by adding specific points with the mouse.

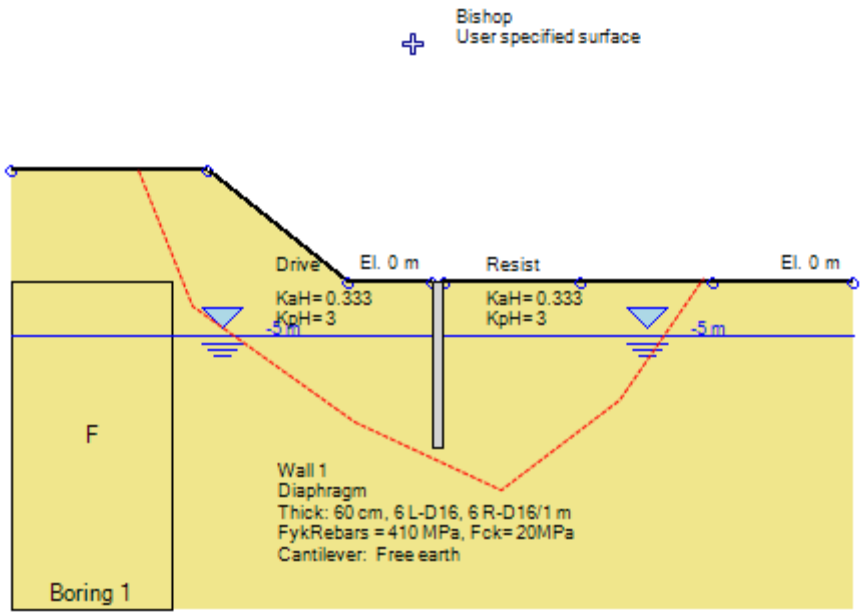


Figure 3.13.5: Custom sliding surface.

- End point limits option (Figure 3.13.6): The user specifies points where the radii for search surfaces pass through (for a failure surface that contains a circular portion).

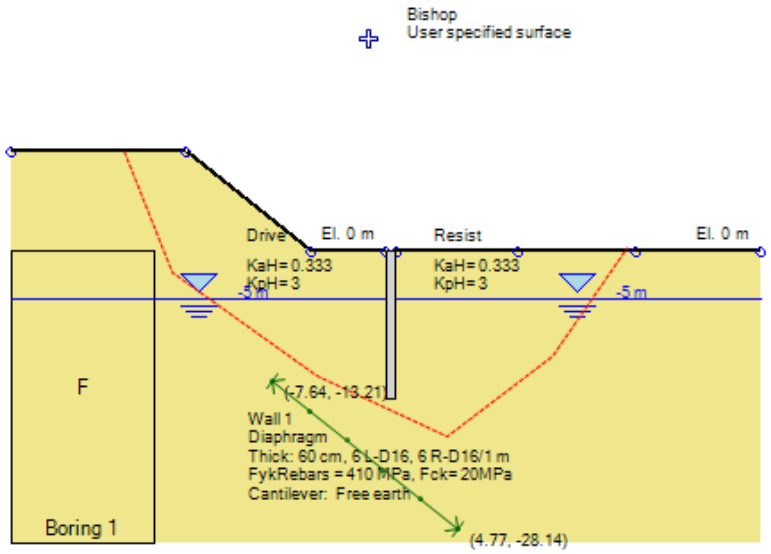


Figure 3.13.6: End point limits.

- Draw the base points for a block (Figure 3.13.7). With this the user can specify 2 points from which the search surfaces pass through for the block type analysis.

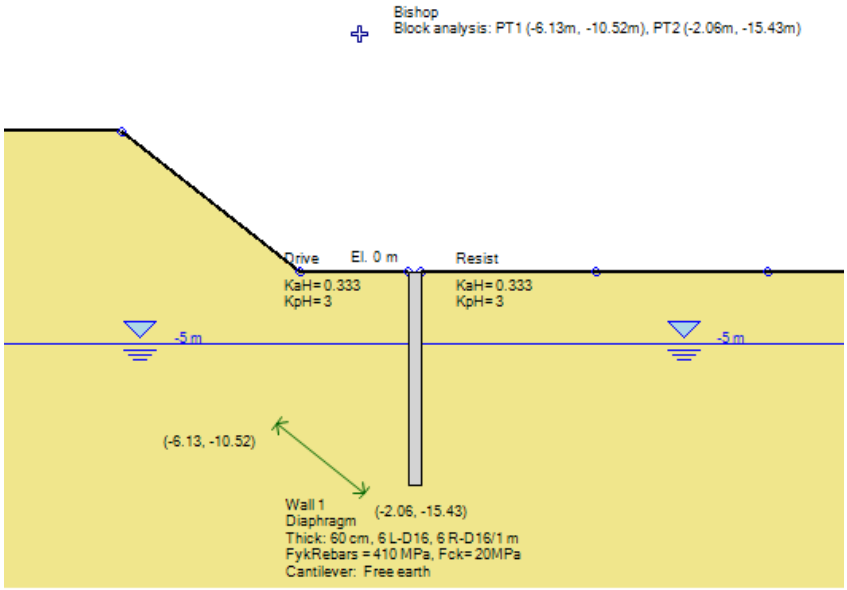


Figure 3.13.7: Two end points.

By clicking on the options button, the slope stability options dialog appears. In the method tab (Figure 3.13.8), we can define the slope stability method used to analyze the model.

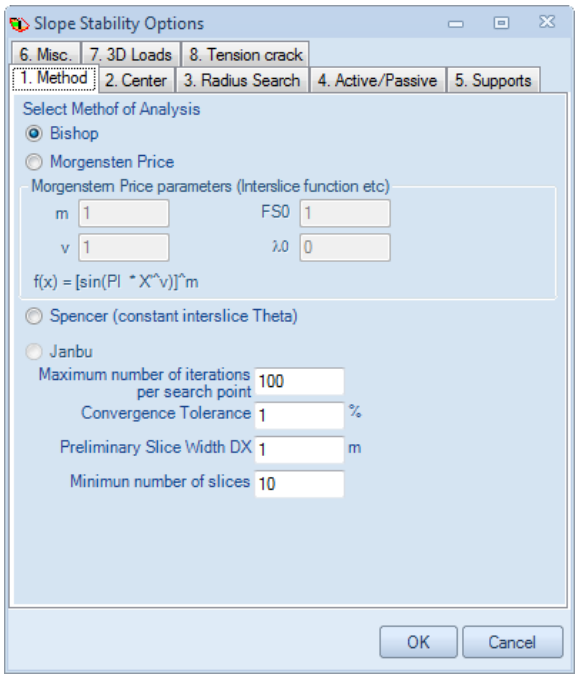


Figure 3.13.8: Slope stability options – method tab.

The following properties are available in this tab:

Symbol	Description
Bishop	Select Bishop simplified method for the slope stability analysis
Morgenstern Price	Select Morgenstern Price method for the slope stability analysis
m	Factor for interslice force equation (if Morgenstern Price method is selected)
v	Factor for interslice force equation (if Morgenstern Price method is selected)
FS0	Initial assumption for factor of safety (if Morgenstern Price method is selected)
λ0	Initial assumption for scaling factor (if Morgenstern Price method is selected)
Spencer	Select Spencer method for the slope stability analysis
Jambu	Select Jambu method for the slope stability analysis
-	Define the maximum number of iterations for each search point
-	Define the convergence Tolerance
DX	Define the preliminary slice width
-	Define the minimum number of slices

In the center tab (Figure 3.13.9), the user can define the properties for the centers of the circles used in the slope stability analysis.

Figure 3.13.9: Slope stability options – center tab.

The following properties are available in this tab:

Symbol	Description
Use only one point for the analysis	Choose to use only one point for the center of the circular type failure
Use exact coordinates for center search	Choose to use exact coordinates for center search. Otherwise coordinates will be relative to the left wall (top corner)
Left	Left search limit (for a rectangular type search)
Right	Right search limit (for a rectangular type search)
steps	Search steps between the two horizontal limits (for a rectangular type search)
Top	Top search limit (for a rectangular type search)
Bottom	Bottom search limit (for a rectangular type search)
steps	Search steps between the two vertical limits (for a rectangular type search)
Rotation	Rotation angle (used for rectangular search)

In the Radius search tab (Figure 3.13.10), we can define the properties for the radius of the circles used in slope stability analysis.

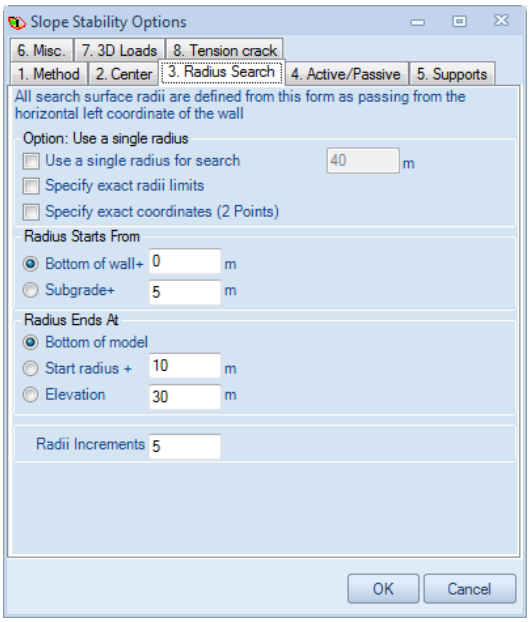


Figure 3.13.10: Slope stability options – radius search tab.

The following properties are available in this tab:

Symbol	Description
Use a single radius for search	Choose to use a single radius for search
Specify exact radii limits	Choose to Specify exact radii limits (for start and end radius)
Initial R	Initial radius limit (if the choice to Specify exact radii limits is selected)
Final R	Final radius limit (if the choice to Specify exact radii limits is selected)
Specify exact coordinates	Choose to search radii with two points
Initial X	Initial search point X-coordinate
Initial Z	Initial search point Z-coordinate
Final X	Final search point X-coordinate
Final Z	Final search point Z-coordinate
Radius starts at	Define the Radius start limits (if no other option is selected)
Radius ends at	Define the Radius end limits (if no other option is selected)
Radii increments	Define the radii increments

In the Active/passive tab (Figure 3.13.11), we can control options for including active and passive wedges for the slope stability analysis.

Figure 3.13.11: Slope stability options – Active/passive tab.

The following properties are available in this tab:

Symbol	Description
Include active angle limit	Choose to include active angle limit and define it
Sweep active angle	Choose to sweep active angle (if the previous option is selected)
Angle Range+ , steps	Define Angle range and steps (if the sweep option is selected)
Include passive angle limit	Choose to include passive angle limit and define it
Sweep passive angle	Choose to sweep active angle (if the previous option is selected)
Angle Range+ , steps	Define Angle range and steps (if the sweep option is selected)
Use block analysis	Choose to use block analysis
Initial X	Initial block point X-coordinate (if the block analysis option is selected)
Initial Z	Initial block point Z-coordinate (if the block analysis option is selected)
Final X	Final block point X-coordinate (if the block analysis option is selected)
Final Z	Final block point Z-coordinate (if the block analysis option is selected)
steps	Define steps for block analysis (if the block analysis option is selected)

In the Supports tab (Figure 3.13.12), the user can define how support forces are included within the slope stability analysis.

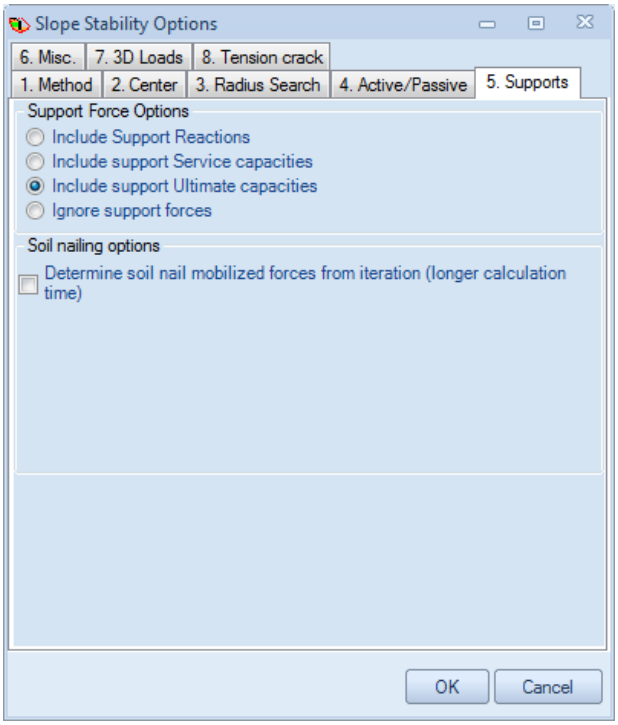


Figure 3.13.12: Slope stability options – Supports tab.

The following properties are available in this tab:

Symbol	Description
Include support reactions	Includes support reactions as analyzed by the wall analysis
Include support service capacities	Includes support service capacities as analyzed by the wall analysis
Include support ultimate capacities	Includes support ultimate capacities as analyzed by the wall analysis
Ignore support forces	Ignores support forces in slope stability analysis
Determine soil nail forces from iteration	Determines mobilized soil nail forces from iteration (when FS = 1)
Define minimum mobilization factor as a percentage of the ultimate capacity	Defines minimum mobilization factor as a percentage of the ultimate capacity (if the previous option is selected)
Inails.mob	Mobilization interaction factor (if the previous option is selected). This factor is used to determine mobilized soil nail forces from: $F_{mob} = (F_{x,ultimate} - F_{x,FS}) \times Inails.mob + F_{x,FS} = 1$

In the Miscellaneous tab (Figure 3.13.13), we can control how the wall shear is accounted in the slope stability analysis (when a wall is intersected).

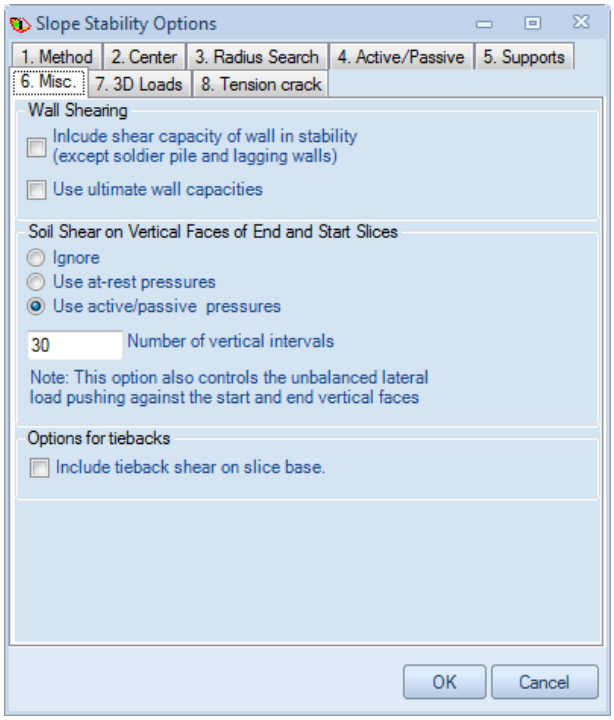

Slope Stability Options
1. Method
2. Center
3. Radius Search
4. Active/Passive
5. Supports
6. Misc.
7. 3D Loads
8. Tension crack
Wall Shearing
Inlcude shear capacity of wall in stability (except soldier pile and lagging walls)
Use ultimate wall capacities
Soil Shear on Vertical Faces of End and Start Slices
Ignore
Use at-rest pressures
Use active/passive pressures
30
Number of vertical intervals
Note: This option also controls the unbalanced lateral load pushing against the start and end vertical faces
Options for tiebacks
Include tieback shear on slice base.
OK
Cancel

Figure 3.13.13: Slope stability options – Miscellaneous tab.

The following properties are available in this tab:

Symbol	Description
-	Choose to include shear capacity of wall in stability
-	Choose to include ultimate wall capacities (design shear x FS.structural)
-	Choose to include support ultimate capacities
-	Choose ignore soil shear on vertical faces (at ends)
-	Choose to use at-rest pressures on a vertical face
-	Define the number of vertical intervals (when a vertical end face is included)
-	Choose to include tieback shear on slice base (if tiebacks used in the model). With this feature, the normal stress and shear is increased when the fixed part of the tieback intersects the base of the slice (Bishop method)

In the 3D Loads tab (Figure 3.13.14), we can define how 3D Loads are accounted in slope stability analysis.

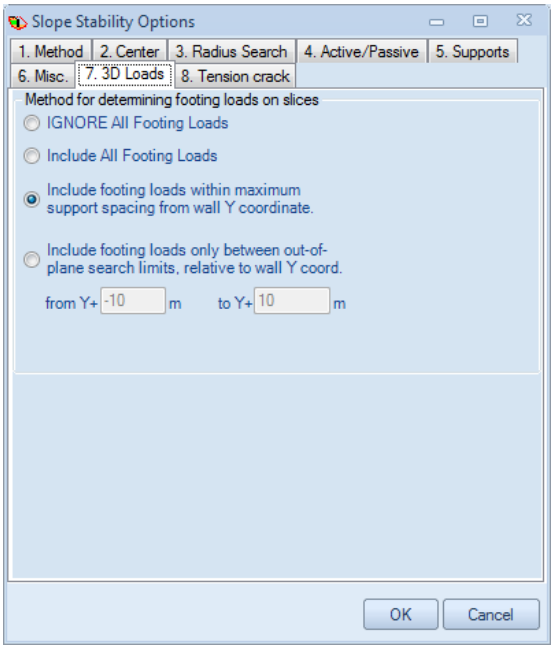


Figure 3.13.14: Slope stability options – 3D Loads tab.

The following properties are available in this tab:

- Ignore all footing loads (and other 3D loads)
- Include all footing loads (and other 3D loads)
- Include footing loads within maximum support spacing from wall Y coordinate
- Include footing loads only between out-of-plane search limits, relative to wall Y coordinate

In the Tension crack tab (Figure 3.13.15), we can control options for including a tension crack within the slope stability analysis.

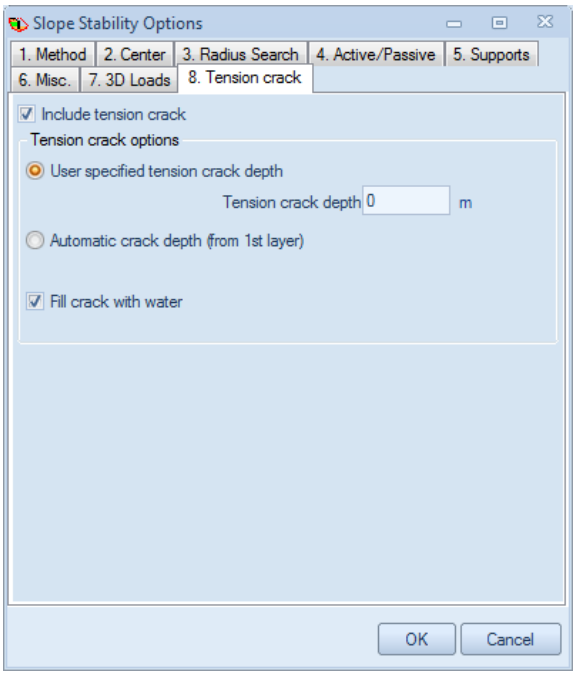


Figure 3.13.15: Slope stability options – Tension crack tab.

The following properties are available in this tab:

- Include tension crack
- Use a specified tension crack depth
- Define the tension crack depth (if the previous option is selected)
- Use automatic crack depth (from top layer)
- Fill the tension crack with water

3.13.2 Soil nailing

Soil nails can be used to reinforce the slope surface and prevent failure. A single soil nail or nail group can be added to the model (Figures 3.13.16 and 3.13.17 respectively). By double-clicking on a single soil nail, the Edit soil nail dialog appears, where we can modify the nail's properties.

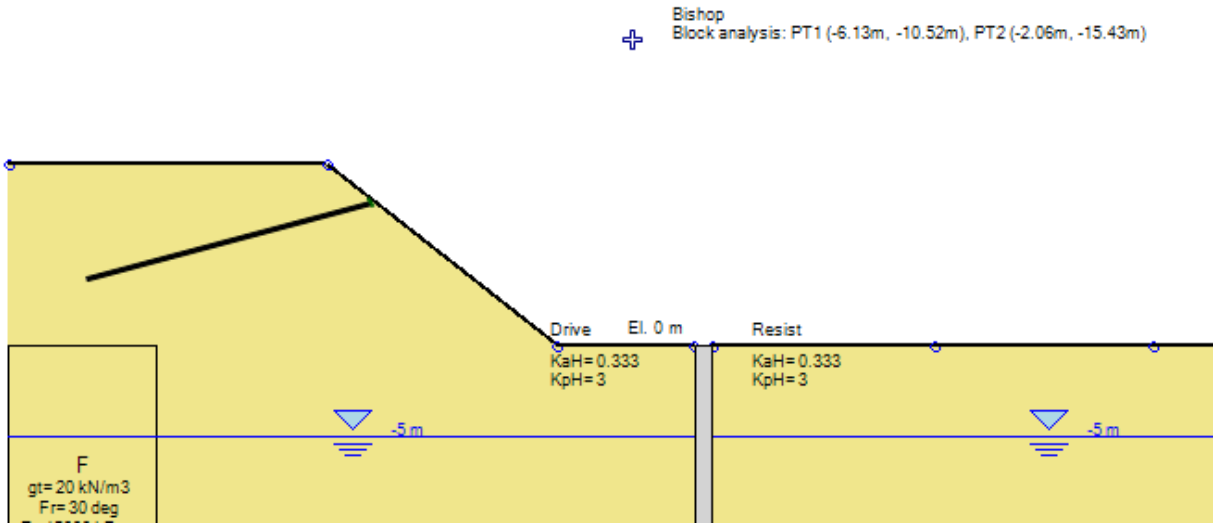


Figure 3.13.16: Add a single soil nail to the model.

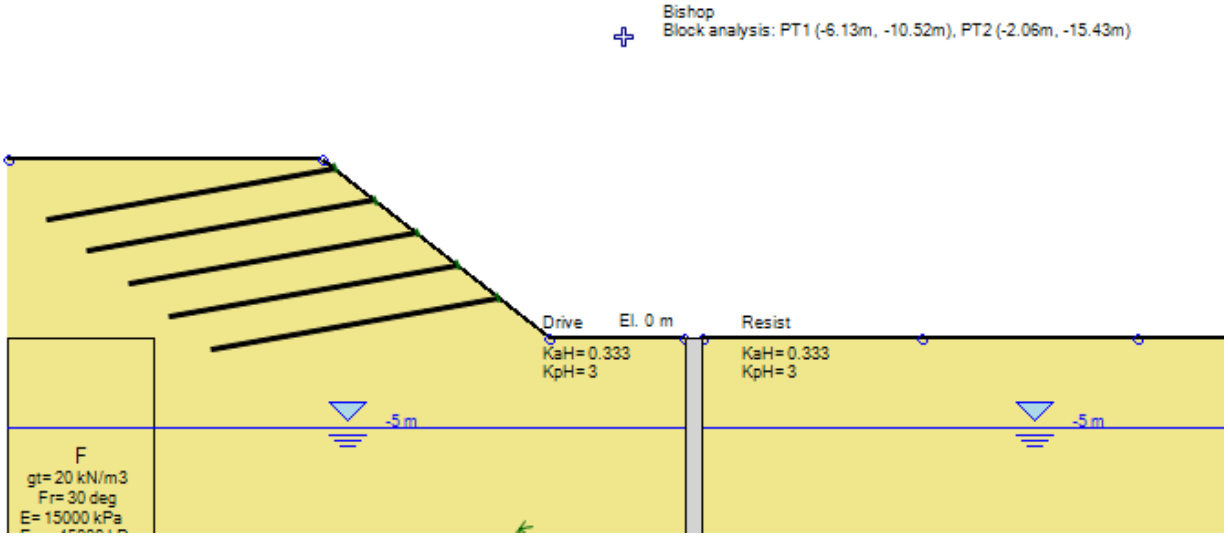


Figure 3.13.17: Add a soil nail group to the model.

In the general tab of the Edit soil nail dialog (Figure 3.13.18) we can set the soil nail's position on the slope, the nail free and fixed length and the nail installation angle. In addition, the user defined the head plate angle and the structural section of the nail can be defined. By pressing the Edit button (Figure 3.13.19) the Tieback Anchors – Sections dialog appears, where the user can choose the strand sections for the soil nail. Next, the user can choose to use capacity loss due to corrosion, as well as some other analysis options (Figure 3.13.20).

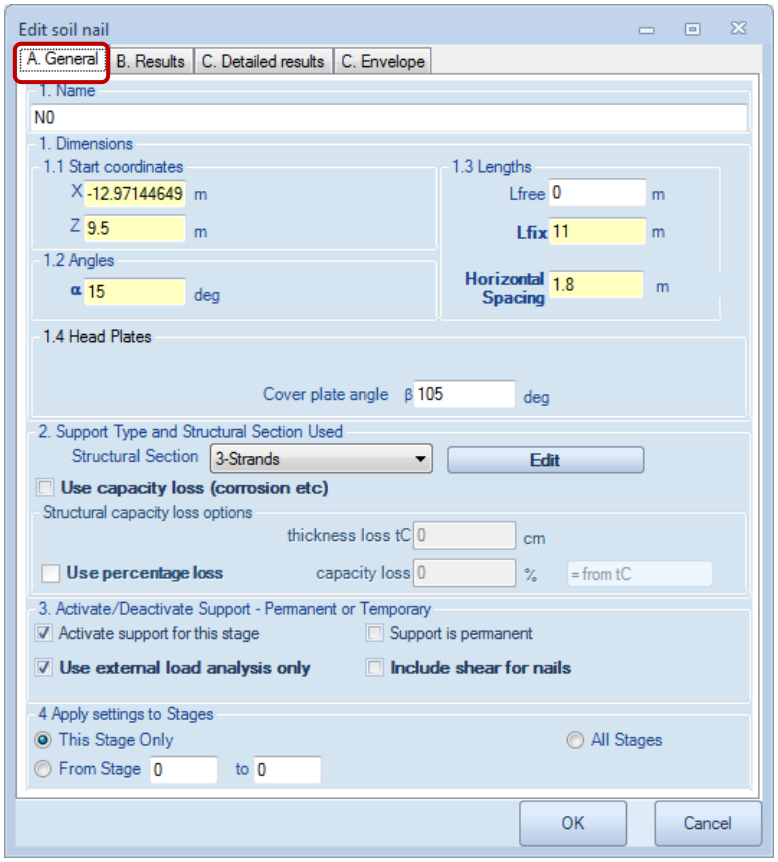


Figure 3.13.18: Edit soil nail dialog.

Symbol	Description
X	The start X-coordinate of the soil nail
Z	The start Z-coordinate of the soil nail
Lfree	The soil nail's free length
Lfix	The soil nail's fixed length
a	The soil nail's installation angle
Horizontal Spacing	The soil nail's horizontal spacing
β	The cover plates angle

The following options are also available:

- Define the structural section of the soil nail and choose to edit it
- Choose to use capacity loss. Then, the user has to define the following properties:
  - ➔ The thickness loss tC, or
  - ➔ The capacity loss as a percentage of the total capacity.
- Choose to activate or deactivate the support in the current stage.
- Define whether the support is permanent or not (affects results when a design approach is used).
- Choose to use only external load analysis for the soil nail.
- Choose to include shear when the nail is intersected by a failure surface.

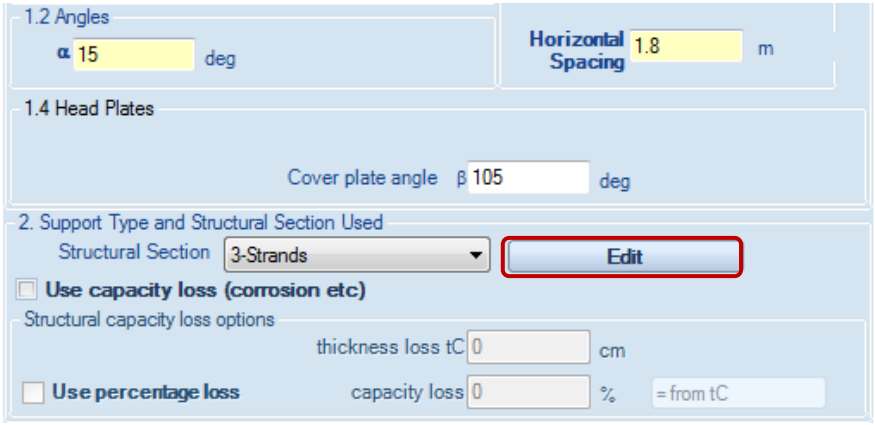


Figure 3.13.19: Edit soil nail section.

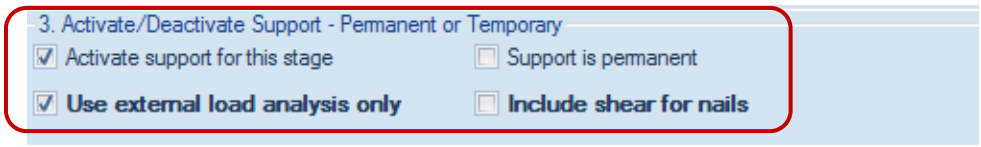

3. Activate/Deactivate Support - Permanent or Temporary
Activate support for this stage
Support is permanent
Use external load analysis only
Include shear for nails

Figure 3.13.20: Analysis options.

In the results tab of dialog (Figure 3.13.21), we can see the following results:

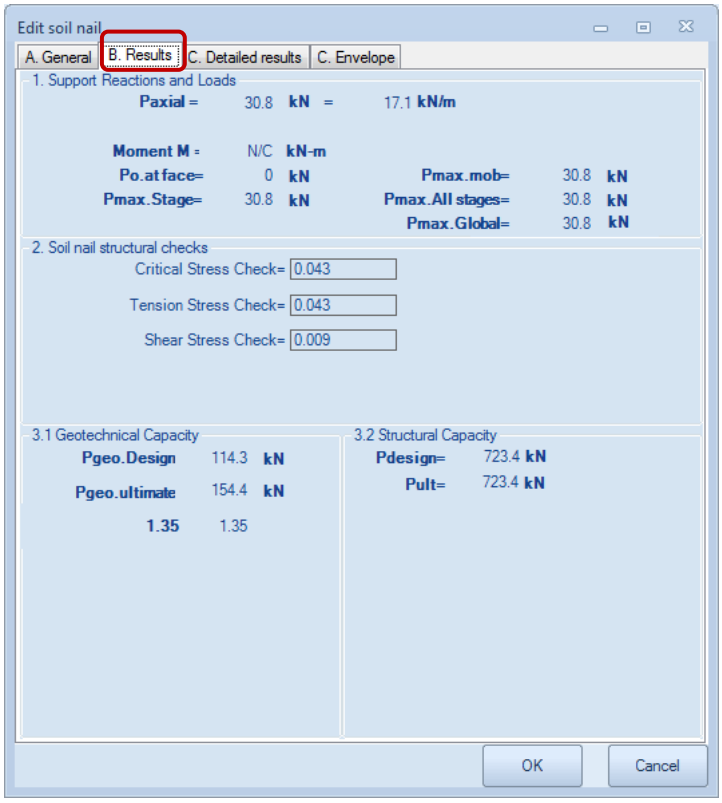


Figure 3.13.21: Results options.

Symbol	Description
Paxial	Axial force of the nail
M	Moment of the nail
Po. At face	Reaction at the facing of the nail
Pmax.mob	The mobilized axial force of the nail
Pmax. stage	Maximum reaction for current stage
Pmax. All stages	Maximum reaction of all stages
Pmax. Global	Maximum reaction of all linked design sections
Pgeo. Design	Geotechnical design capacity
P geo. ultimate	Geotechnical ultimate capacity
Pdesign	Structural design capacity
Pult	Structural ultimate capacity

In the detailed results tab of the Edit soil nail dialog (Figure 3.13.22) we can see the following results in a tabulated format:

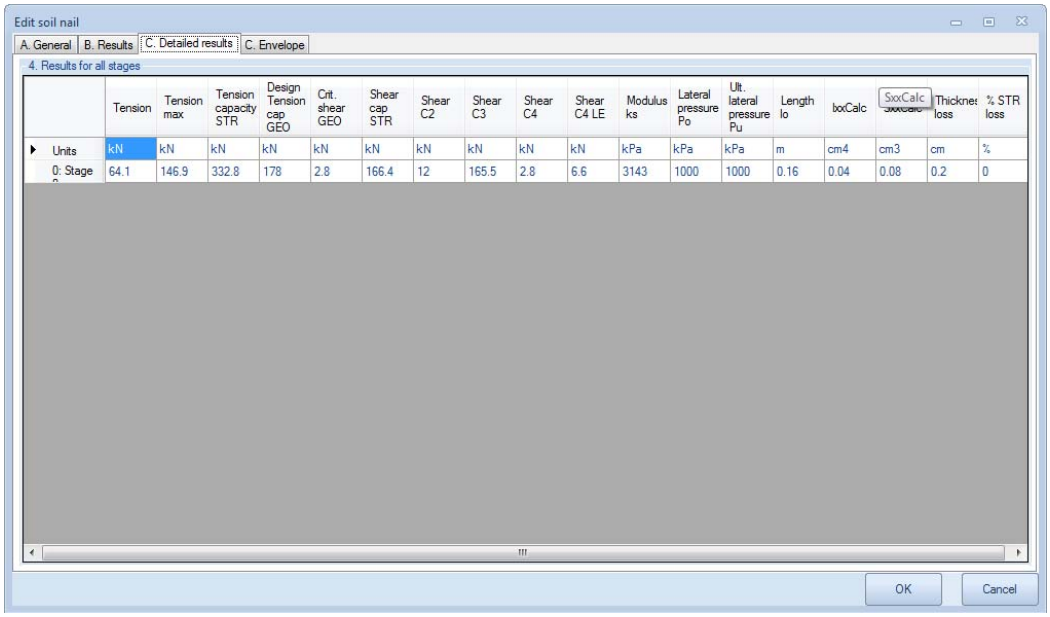


Figure 3.13.22: Detailed results options.

- **Tension:** This column contains the tension of the soil nail because of the critical slope surface.
- **Tension max:** This column contains the maximum tension that occurs to this soil nail because of all slope surfaces that intersect this nail.
- **Tension Capacity STR:** This column contains the geotechnical (pullout) capacity of the soil nail.
- **Critical Shear GEO:** This column contains the minimum shear from those calculated from the three criteria (see below).
- **Shear Capacity STR:** This column contains the structural shear capacity.
- **Shear C2:** This column contains the shear as calculated from the three criteria (C1, C2, C4) (see slope stability manual, pages 31 – 33).
- **Shear C3:** Same as above.
- **Shear C4:** Same as above.
- **Shear C4 LE:** Same as above (LE: Limit equilibrium analysis)

- **Modulus ks:** This column contains the lateral subgrade reaction modulus at the critical intersection surface point.

- **Lateral pressure:** See slope stability manual, page 31.

- **Ultimate lateral pressure:** See slope stability manual, page 31.

- **Length:** This column includes the calculated transfer length of the soil nail at the intersection with the critical slope surface.

- **Ixx calc:** This column includes the calculated moment of inertia of the nail (includes all reductions for corrosion etc).

- **Sxx calc:** This column includes the section modulus (European designation is W_{el})

- **Thickness loss:** This column includes the soil reinforcement thickness loss.
- **% STR loss:** This column includes the percentage of structural capacity loss (when corrosion is included).

3.14 Data entry: Wale beams

In DeepXcav, it is possible for to add walers to supports. DeepXcav calculates the capacity and the loads of the installed walers. Wale beams are added at the connection points of the supports with the walls. In order to install a new waler, we first have to install a support. Next, by pressing the Wales button

in the Loads/supports menu and choosing the support, the Wales dialog appears (Figure 3.14.1). In this area we can choose the loading type of the waler. Several loading cases are included, using point or uniform loads. These cases are presented in the following table.

In this dialog the user can:

- Edit the waler name.
- Define the section used for the waler and edit it.
- Define the vertical spacing of the waler amongst:
  - ➔ Use support spacing
  - ➔ Use wall spacing
  - ➔ Use a user defined value
- Define the axial force amongst:
  - ➔ Use zero axial force
  - ➔ Use as axial force a percentage of support reactions
  - ➔ Use a user defined value
- Define the loading type:
  - ➔ Use point loads (ideal for soldier pile walls)
  - ➔ Use uniform loads

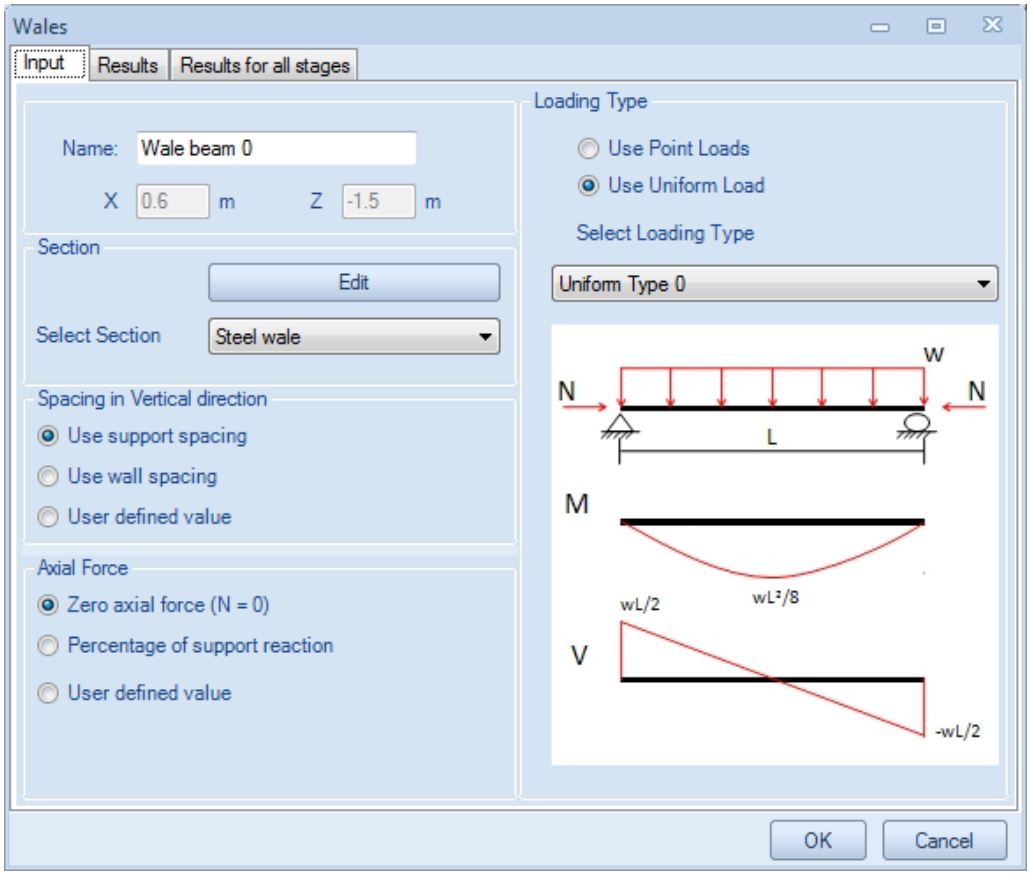
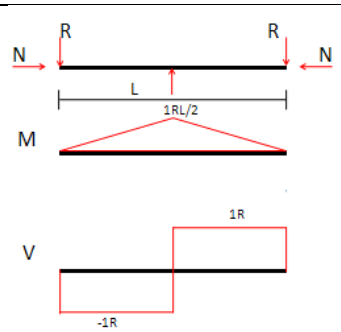
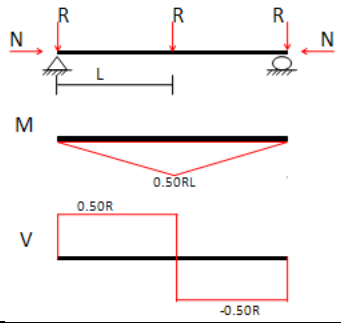
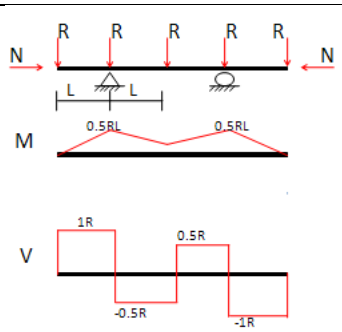
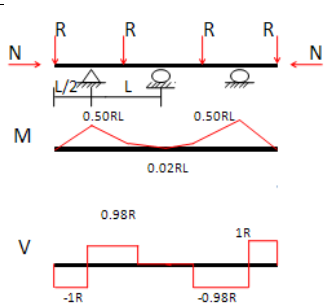
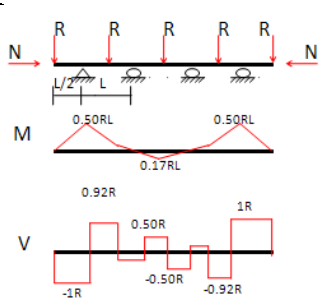
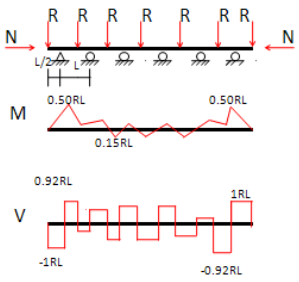
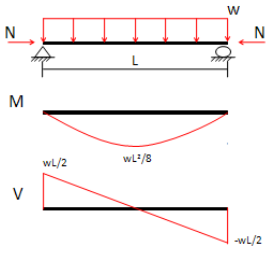
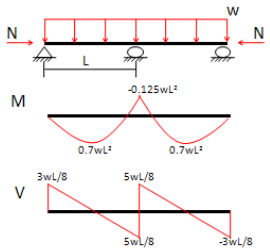
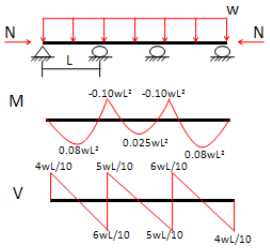
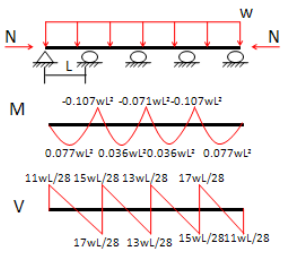
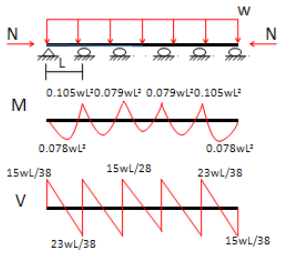
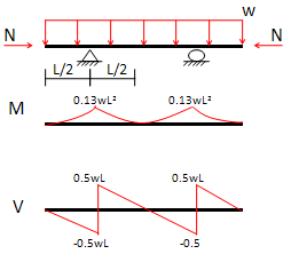
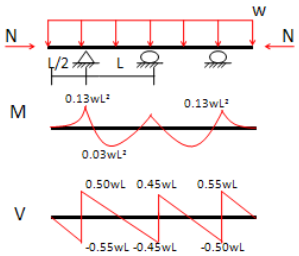
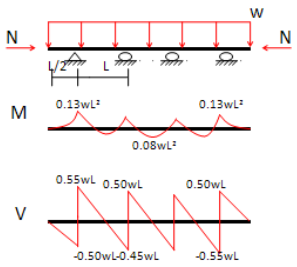
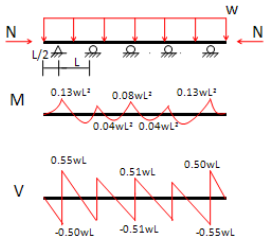


Figure 3.14.1: The waler dialog.

Point load type 0	
Point load type 1	

Point load type 2	
Point load type 3	
Point load type 4	
Point load type 5	
Uniform type 0	

Uniform type 1	
Uniform type 2	
Uniform type 3	
Uniform type 4	
Uniform type 5	
Uniform type 6	

Uniform type 7	
Uniform type 8	

Next, we can choose the wale section by pressing the Edit button (Figure 3.14.2). The user can select among using a steel or a concrete section waler (Figures 3.14.3 and 3.14.4 respectively).

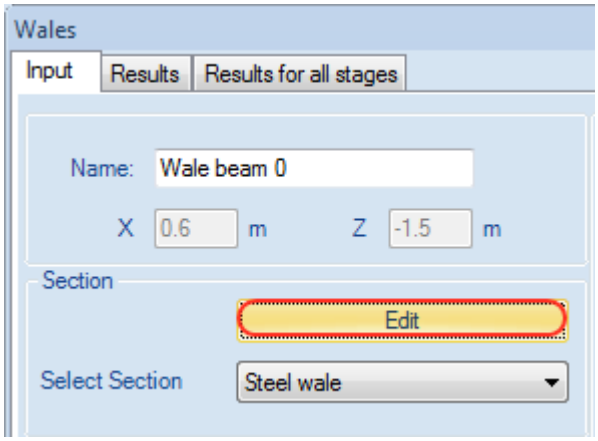


Figure 3.14.2: The edit waler section button.

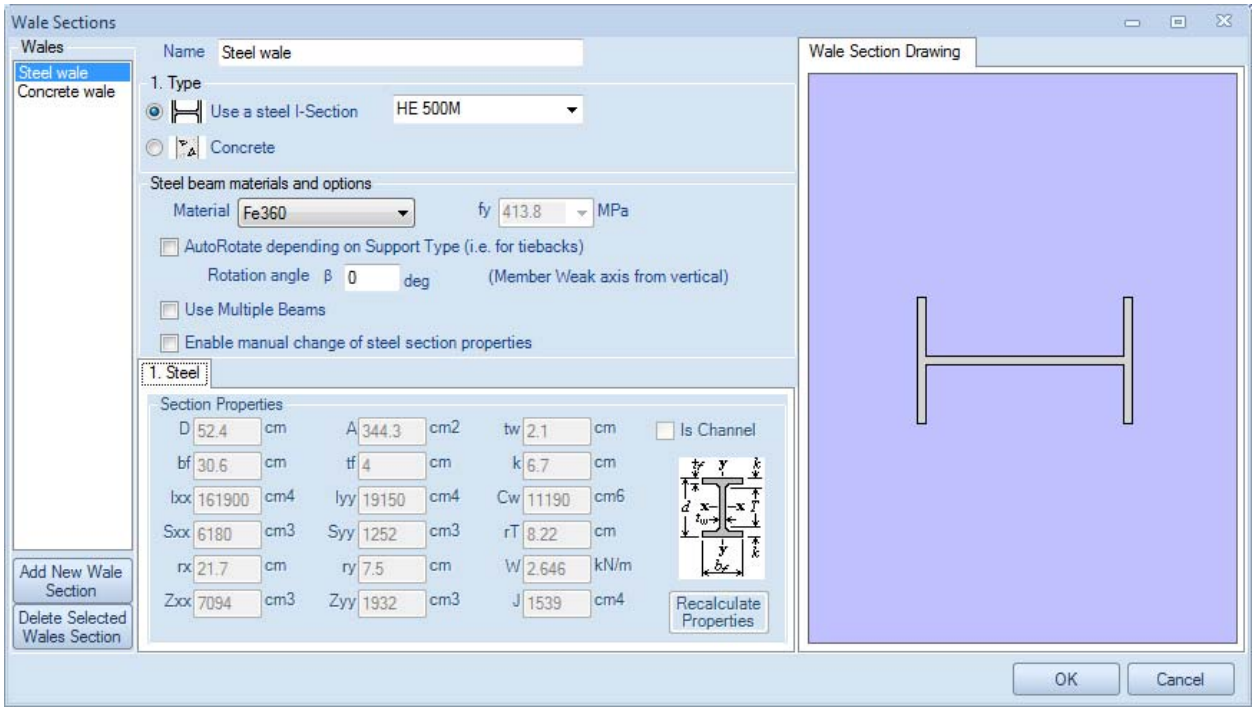


Figure 3.14.3: Steel section.

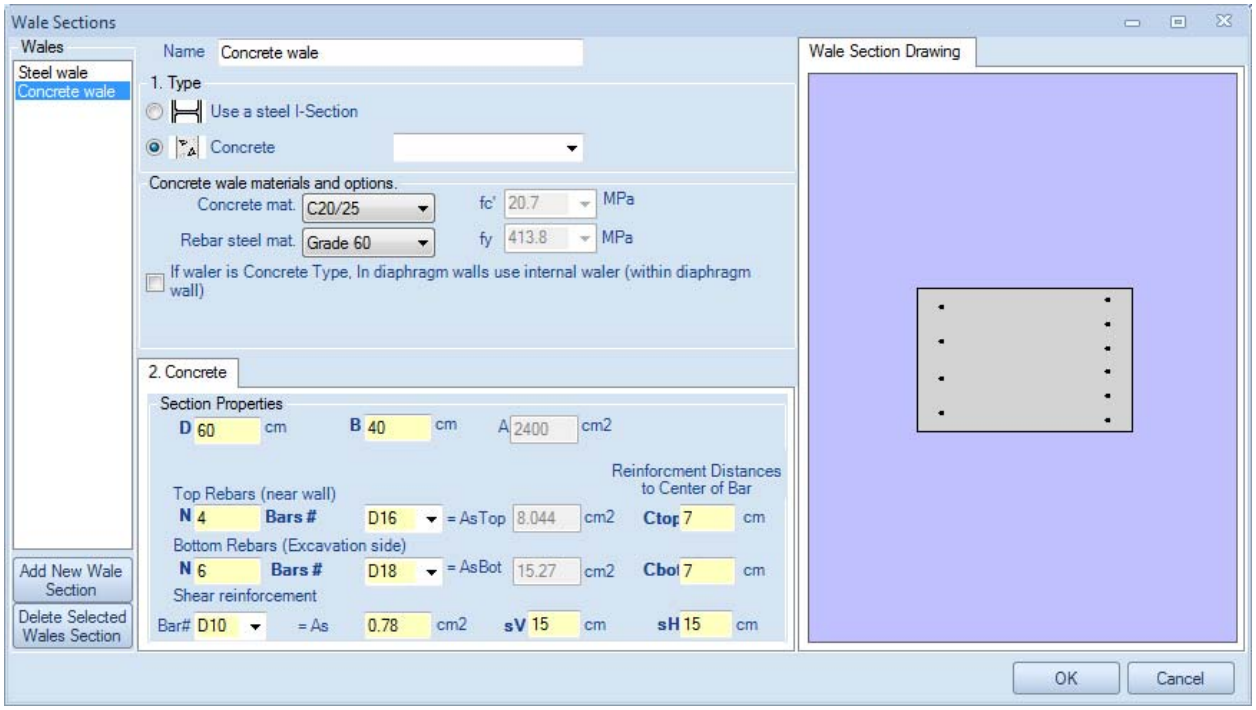

Wale Sections
Wales
Steel wale
Concrete wale
Name Concrete wale
1. Type
Use a steel I-Section
Concrete
Concrete wale materials and options.
Concrete mat. C20/25
fc' 20.7 MPa
Rebar steel mat. Grade 60
fy 413.8 MPa
If waler is Concrete Type, In diaphragm walls use internal waler (within diaphragm wall)
2. Concrete
Section Properties
D 60 cm
B 40 cm
A 2400 cm2
Reinforcment Distances to Center of Bar
Top Rebars (near wall)
N 4 Bars # D16 = AsTop 8.044 cm2 Ctop 7 cm
Bottom Rebars (Excavation side)
N 6 Bars # D18 = AsBot 15.27 cm2 Cbot 7 cm
Shear reinforcement
Bar# D10 = As 0.78 cm2 sV 15 cm sH 15 cm
Add New Wale Section
Delete Selected Wales Section
Wale Section Drawing
OK
Cancel

Figure 3.14.4: Concrete section.

By choosing to use a steel section, the following options are available:

- Use multiple beams: With this selection, more than one beams can be used to the same support. These beams can be used with a clearance or without one.
- Autorotate the waler: This is possible for some kinds of supports (tiebacks, spring supports), and only if more than one beams are used. The waler can be installed using the support's angle (Autorotate), or the user can define a custom angle (Figure 3.14.5). For other kinds of supports, the waler is installed horizontally (Figure 3.14.6). Rotation is used only for steel section beams.

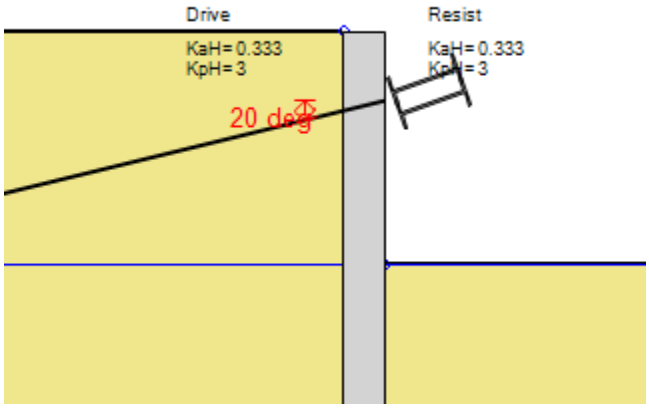


Figure 3.14.5: Rotated wale beam, used with a tieback.

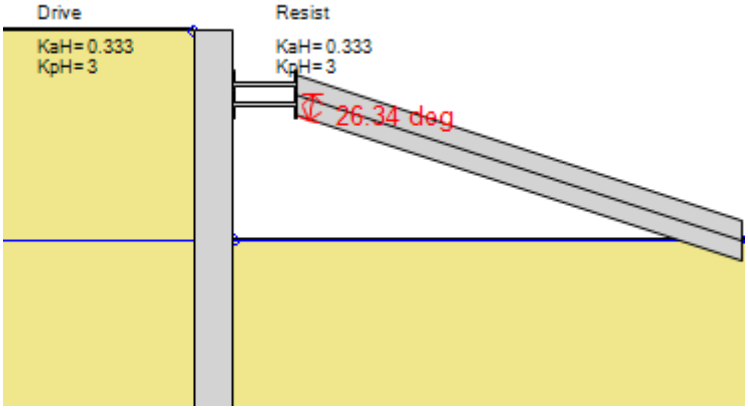


Figure 3.14.6: Horizontal wale beam, used with a raker.

- Enable manual change of steel section properties: With this option, the user can modify manually the waler's properties (the steel beam properties and options are the same as described in paragraph 3.7.1.).

By choosing to use a concrete section (Figure 3.13.7), the user has to define the reinforcement properties. These properties are presented in the table below.

Symbol	Description
D	Waler height (thickness)
A	Area of section
B	Waler base dimension
N	Number of bars
Bar#	Reinforcement bars
As, top	Top reinforcement area
As, bot	Top reinforcement area
Ctop	Top clearance (to center of top bar)
Cbot	Bottom clearance (to center of bottom bar)
sV	Vertical spacing for shear reinforcement
sH	Horizontal spacing for shear reinforcement

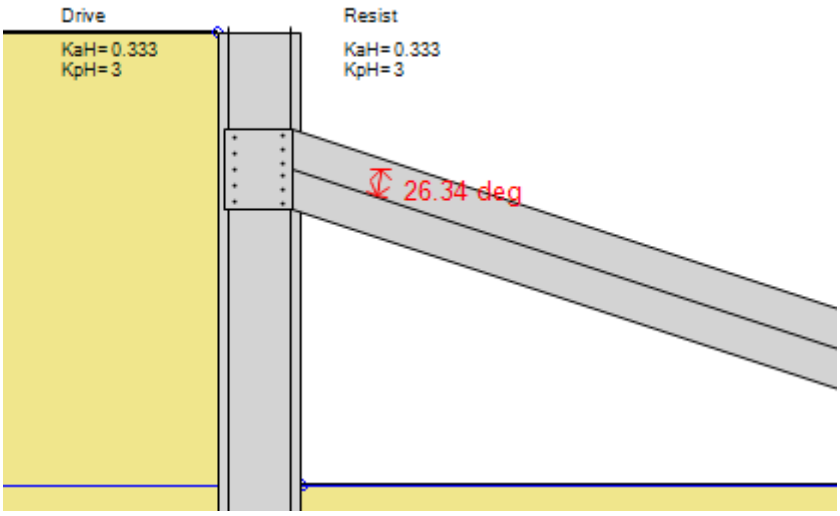


Figure 3.14.7: Concrete section beam, used with a raker.

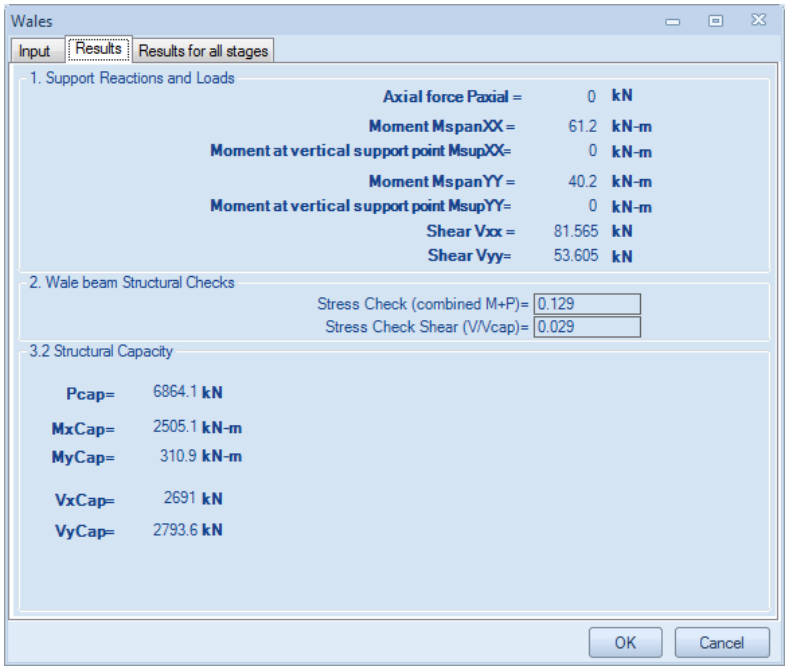


Figure 3.14.8: The waler dialog - results.

Symbol	Description
Paxial	Axial force on the waler
Mspan XX	Span moment along X-axis
Msup XX	Moment along X-axis at support location
Mspan YY	Span moment along Y-axis
Msup YY	Moment along Y-axis at support location
Vxx	Shear along X-axis
Vyy	Shear along Y-axis
Pcap	Axial capacity
Mx cap	Moment capacity along X-axis
My cap	Moment capacity along Y-axis
Vxcap	Shear capacity along X-axis
Vycap	Shear capacity along Y-axis

CHAPTER 4: MODIFYING MODELS & VIEWING RESULTS

4.1 Modifying surface elevations

DeepXcav includes powerful features and versatile options for modeling surface elevations and sections. Elevations and points can be modified with the following options:

- Double click on a surface point: This gives the possibility to modify the point's coordinates (Figure 4.1.1).

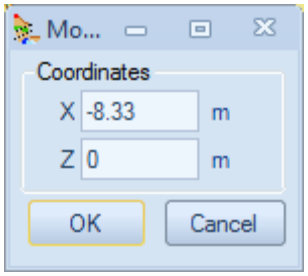


Figure 4.1.1: Modify single point's coordinates.

- Select to set a bench or slope surface (Figures 4.1.2 and 4.1.3 respectively), by clicking on the right selection at the Surface options button of the Model menu (Figure 4.1.4), or by right clicking on the model interface (Figure 4.1.5). The forms that create the slope or bench surfaces are presented in paragraph 2.6.

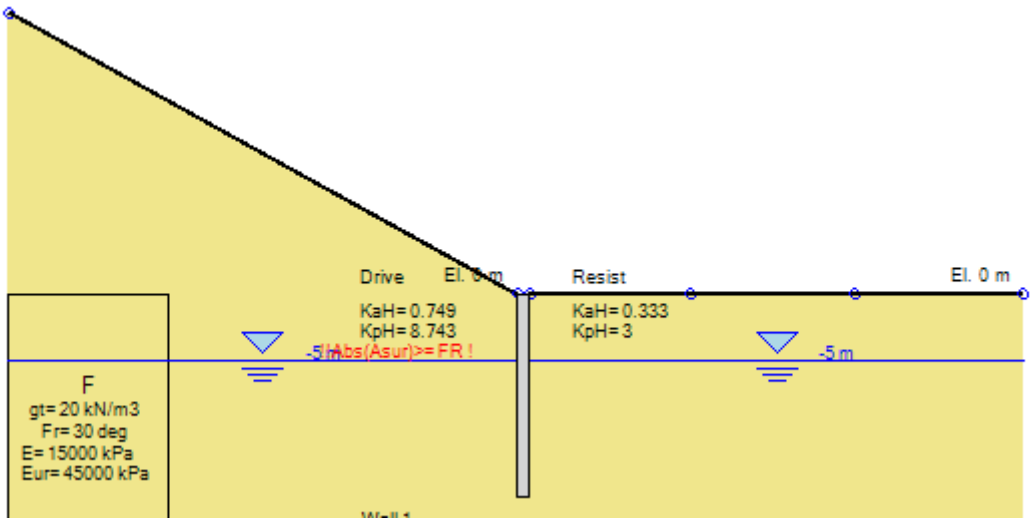


Figure 4.1.2: Left slope surface.

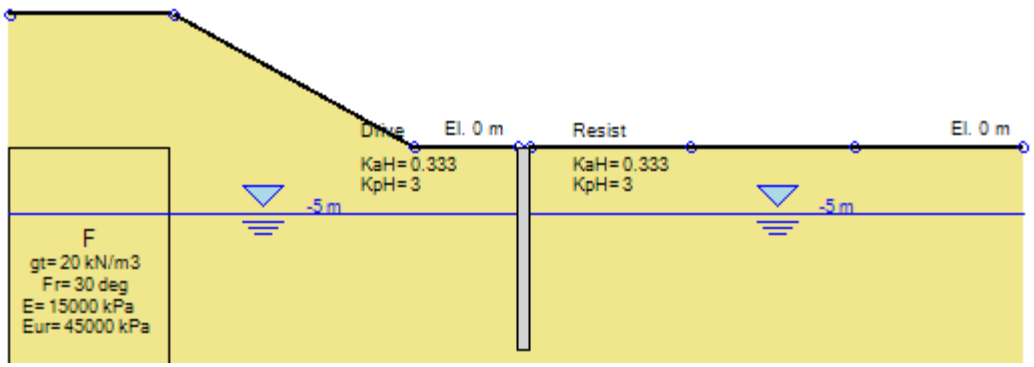


Figure 4.1.3: Left slope surface.

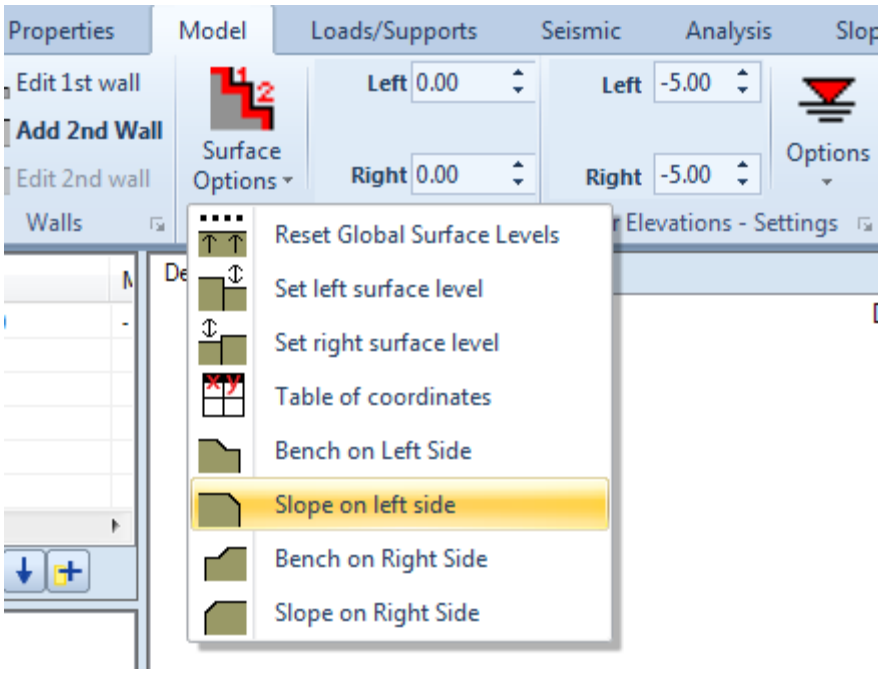


Figure 4.1.4: Surface options.

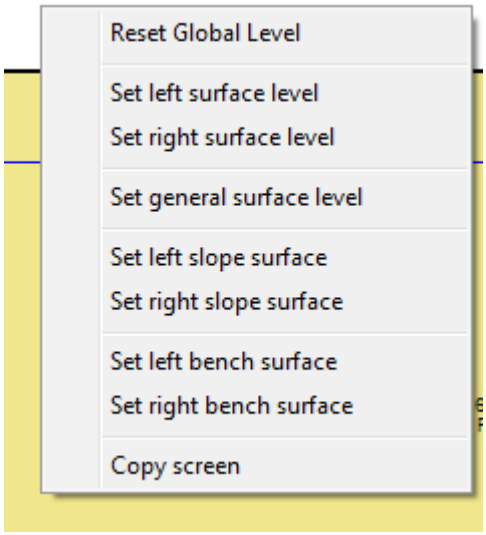


Figure 4.1.5: Right click on the model.

The following options are available:

Icon	Description
	Resets global surface levels
	Sets right surface level
	Sets left surface level
	Table of coordinates of surface points
	Create a bench on the left side
	Create a slope on the left side
	Create a bench on the right side
	Create a slope on the right side
	Resets global surface levels
	Sets right surface level
	Sets left surface level

- By pressing the Table of coordinates selection of the surface options (Figure 4.1.4), the Table of coordinates dialog appears (Figure 4.1.6). This dialog contains the coordinates of all surface points. The user can then modify them here, and add or remove points within.

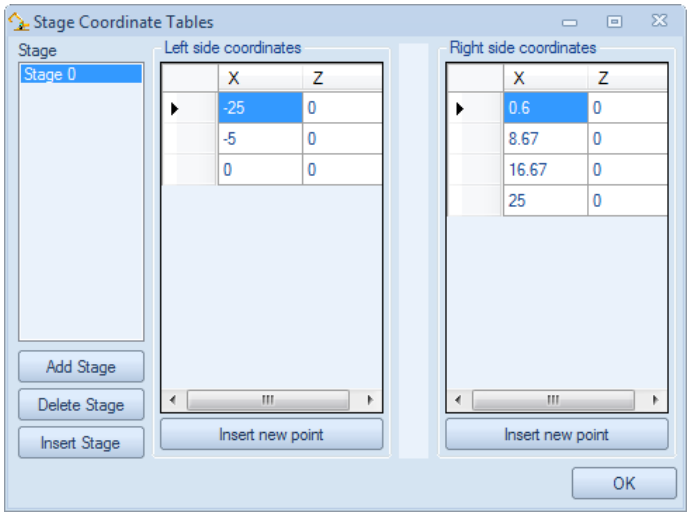


Figure 4.1.6: Table of coordinates dialog.

- We can perform an excavation or a backfill operation from the icons of the toolbar at the left side of the screen (see paragraph 2.3.1). Then, by clicking on several points on the screen, the user can perform these operations (Figures 4.1.6 and 4.1.7). The backfill or excavation can be completed by pressing enter.

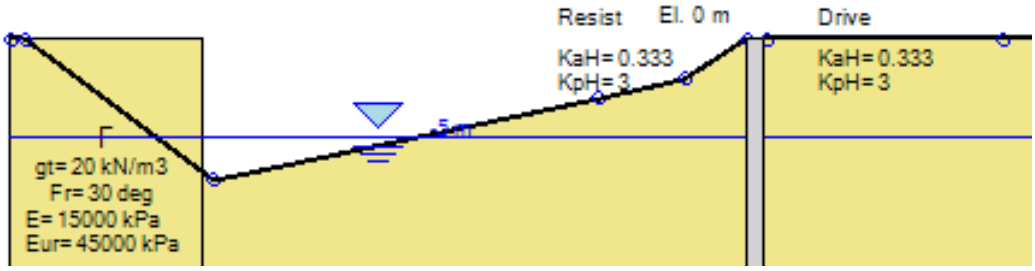


Figure 4.1.6: Excavation operation using the Excavation operation tool.

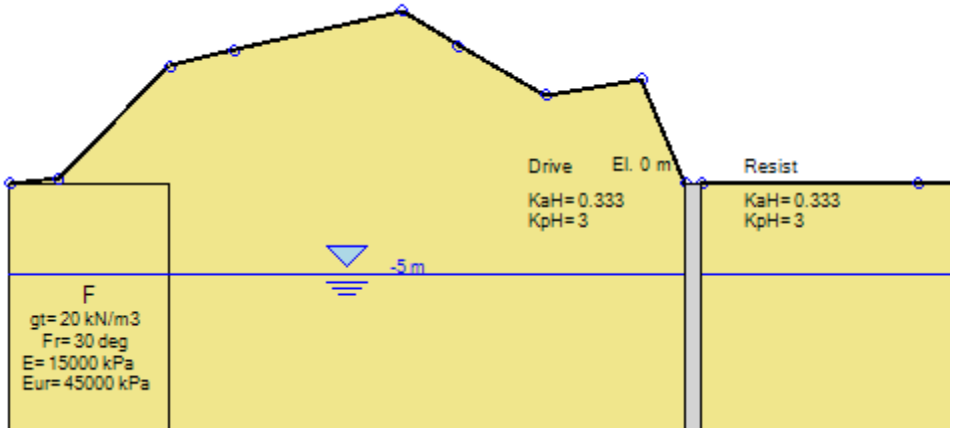


Figure 4.1.7: Backfill operation using the Backfill operation tool.

4.2 Adding supports graphically

Supports in DeepXcav can be added graphically, that is the user has to draw the support first by selecting the appropriate tool. Instructions, on specific supports are provided in the following table.

Toolbar Icon	Description
	Adds a tieback support (you have to click at the main stage form to graphically add the tieback). A tieback can also be used to connect two walls. In this case the tieback will be treated as a tie rod.
	Adds a strut support (you have to click at the main stage form to graphically add the support). Struts can also be used to connect two opposite walls.
	Adds a raker support. DeepXcav treats rakers as inclined struts. WARNING: USER MUST ENSURE THAT THE RAKER FALLS OUTSIDE PASSIVE ZONE OF INFLUENCE FOR RAKERS TO BE PRACTICALLY EFFECTIVE. DEEP DOES NOT AUTOMATICALLY ADJUST RAKER REACTIONS IF A RAKER FALLS WITHIN THE PASSIVE ZONE. HOWEVER, FOR SLOPE STABILITY, DEEP WILL IGNORE A RAKER IF IT ENTIRELY FALLS WITHIN THE SEARCH SURFACE (OR CIRCLE)

	Adds a slab support (you have to click at the main stage form to graphically add the support). Slabs can also be used to connect two opposite walls.
	Adds a fixed support (you have to click at the main stage form to graphically add the support).
	Adds a spring support(you have to click at the main stage form to graphically add the support).

Example: Add a Tieback

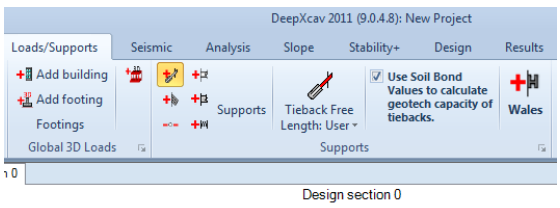
Step 1: Click on Add a tieback Icon

Step 2: Click location on wall (Figure 4.2.1)

Step 3: Click second point of tieback (end point in soil).

Step 4: The Support Edit dialog appears. Refine the new tieback support and click the OK button.

The support is successfully inserted to the model (Figure 4.2.2). To change the supports properties, double click on the support, or select it from the tree menu.



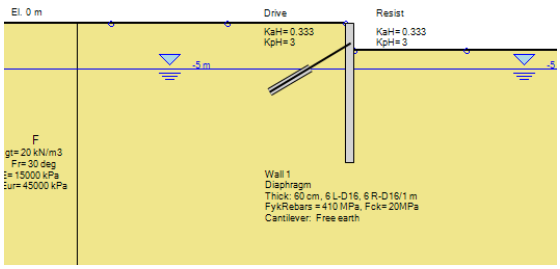
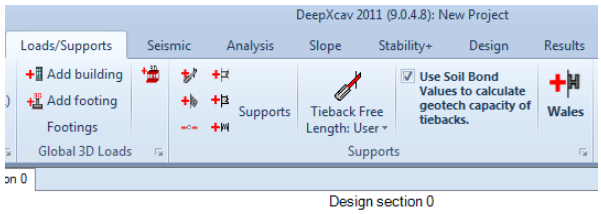


Figure 4.2.1: Click on the wall (step 2).



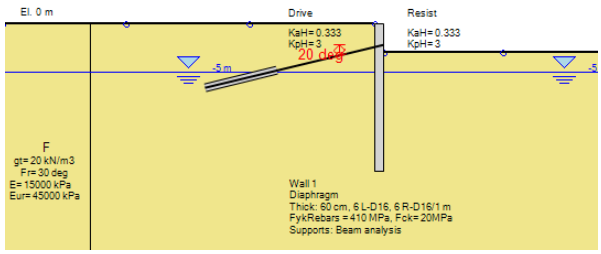


Figure 4.2.2: Placed tieback.

The following figures present a tieback connecting two walls, a slab and a strut support. To install these supports, the steps of the previous example have to be followed.

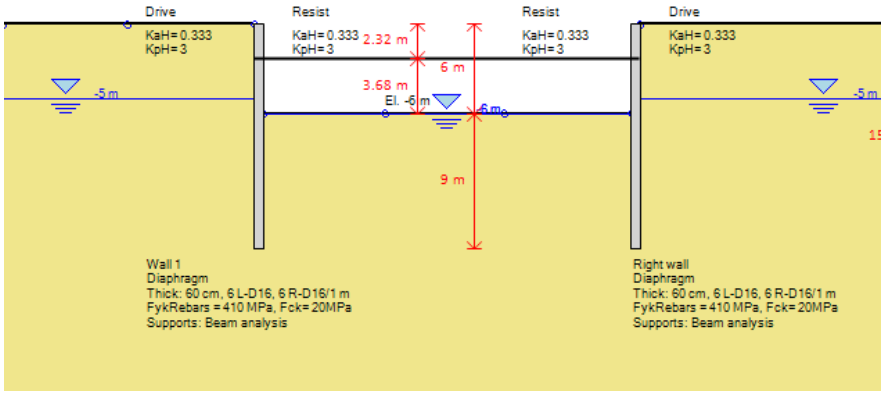


Figure 4.2.3: A tieback connecting two walls.

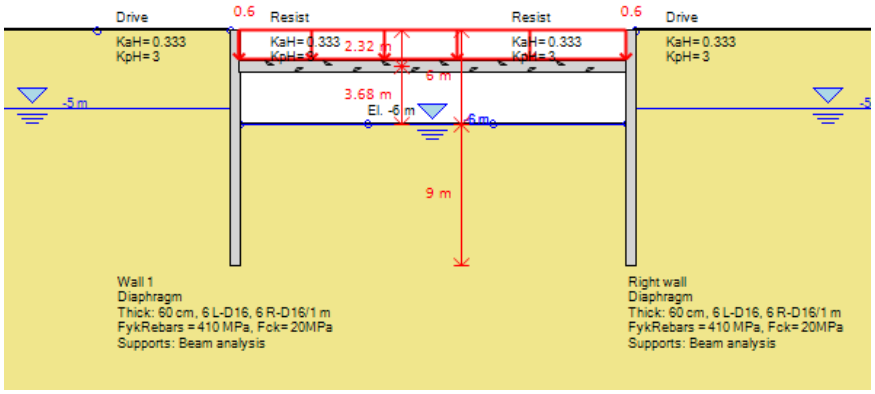


Figure 4.2.4: A slab connecting two walls.

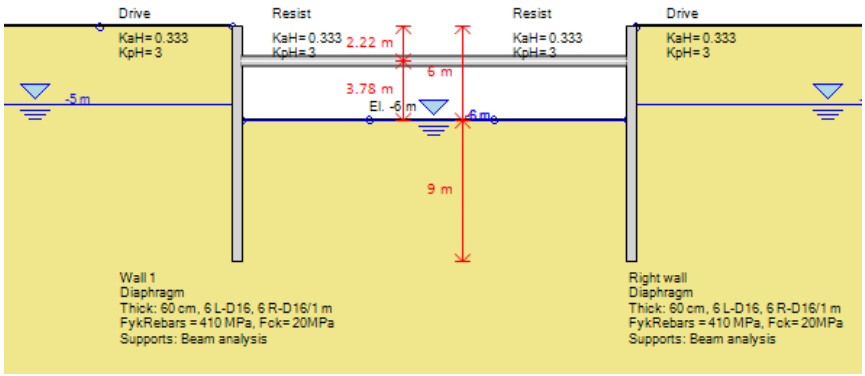


Figure 4.2.4: A strut connecting two walls.

4.3 Adding Surcharges Graphically

Surcharges and line loads in DeepXcav can be added graphically. The user has to draw the surcharge first by selecting the appropriate tool. Instructions, on specific supports are provided in the following table. DeepXcav uses equations by DM7.2, Poulos & Davis, and Kelvin to calculate the horizontal surcharge stress q on the wall. The methods and are most appropriate for loads on horizontal surfaces, but is most cases yield satisfactory results for non level surfaces. DeepXcav integrates the horizontal and the vertical component of each surcharge. You can vary the load magnitudes for the same surcharge for different stages.

	Adds a *Surface Surcharge.* A *Surface surcharge* is a strip surcharge that can have its elevation changed for different stages (automatically adjusted to the surface elevation) but has always the same horizontal coordinates. The surcharge can have both a horizontal and a vertical component. You need to define a surcharge by two points (this creates a default 1.0 kPa load). Then in order to modify the surcharge, double click on it and the **Edit Distributed Load** dialog appears.
	Adds a *Surface Line Load*. Surface line loads are treated in the same way as surface surcharges. To modify a Surface Line Load, double click on it.
	Adds a *Wall Surcharge.* A *Wall surcharge* is a strip surcharge applied directly on the wall. You need to define a wall surcharge by clicking on two points on the wall (this creates a default 1.0 kPa load). Then in order to modify the surcharge, double click on it and the Edit Surcharge dialog appears.
	Adds a *Wall Line Load*. Surface line loads are treated in the same way as wall surcharges. To modify a Surface Line Load, double click on it.
	Adds a three dimensional footing load. Once you select the button then you must click on the main form at the location where you want the center of the footing to be. Then the edit footing form automatically launches and you can specify all the parameters for this footing load.

Example: Adding a Surcharge

Step 1: Click on Add a Surface Surcharge icon **(Figure 4.3.1)**
Step 2: Click first location for surcharge (Figure 4.3.2)
Step 3: Click second location for surcharge (Figure 4.3.3)
Step 4: Double click on new surcharge at form to launch the Surcharge Edit menu and refine the new surcharge.
The surcharge is successfully inserted to the model (Figure 4.3.4). To change the supports properties, double click on the support, or select it from the tree menu.

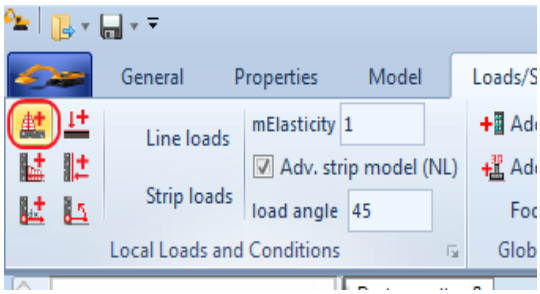


Figure 4.3.1: Add a surcharge tool (step 1).

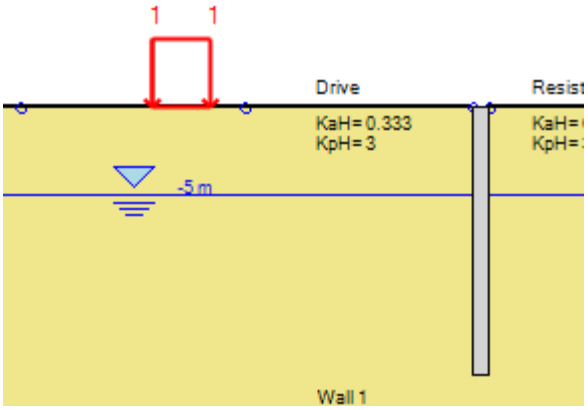


Figure 4.3.2: Select first point (step 2).

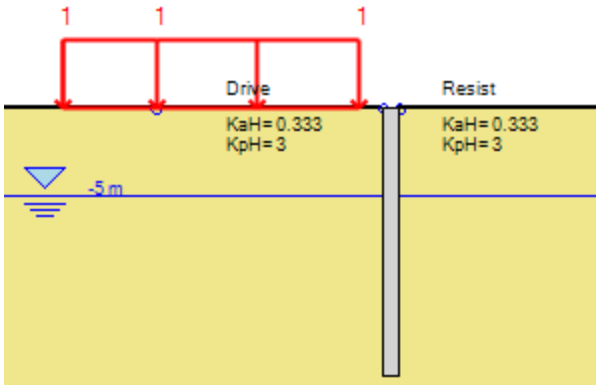


Figure 4.3.3: Select end point (step 3).

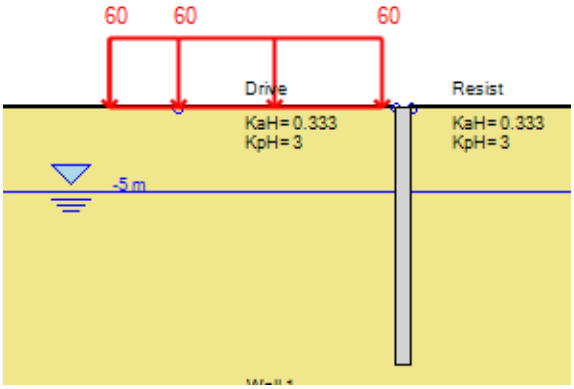


Figure 4.3.4: Placed surcharge

4.4 Viewing Results on Main Form

Once a project is analyzed, results can be viewed on screen by selecting one or more of the Toolbar Icons shown below. Full icon description and associated examples are also shown in the following Exhibits.

Icon	Description
	Shows the Wall embedment Safety Factors
	Shows the Global Slope Factor of Safety
	Shows the Global Slope Factor of Safety Contours
	Shows the results for slope stability slices
	Shows all slope stability results
	Draws the bending moment diagram of the wall on the main model screen.
	Draws the shear force diagram of the wall on the main model screen.
	Draws the axial force diagram of the wall on the main model screen.
	Draws the wall deflections on the main model screen.
	Draws the Surface settlements on the main model screen.
	Shows the support reactions on the main model screen.
	Shows the combined capacity ratio (Moment and Axial force) on the main model screen.
	Shows the moment capacity ratio on the main model screen.
	Shows the shear capacity ratio on the main model screen.
	Shows the structural capacity on the main model screen.
	Shows the wall moment capacity on the main model screen.
	Shows the wall shear capacity on the main model screen.

	Draws the effective horizontal soil pressure on the main model screen.
	Draws the total vertical stress on the main model screen.
	Draws the effective vertical stress on the main model screen.
	Draws the net total wall pressures on the main model screen.
	Draws the wall surcharge on the main model screen.
	Draws the seismic pressures on the main model screen.
	Draws the net water pressure on the main model screen.
	Draws the water pressure on walls
	Hydraulic gradient
	Shows Minimum – Maximum values on the diagrams.
	Shows envelope of moments and support reactions when moments or support reaction results are displayed.
	Loads the result tables.
	Shows the water pressure shadings.
	Draws the water head shadings.
	Draws the water head grid.
	Shows the hydraulic safety factor.
	Draws the total vertical stress shadings
	Draws the effective vertical stress shadings

The following figures show some typical on screen output diagrams and results. Output results can be visible only if the given problem has been analyzed. Feel free to explore the functionality of these toolbars. DeepXcav can stack the result diagram axes so this way multiple results can be visible at the same time for direct comparison. However, you should try to keep the output as simple as possible.

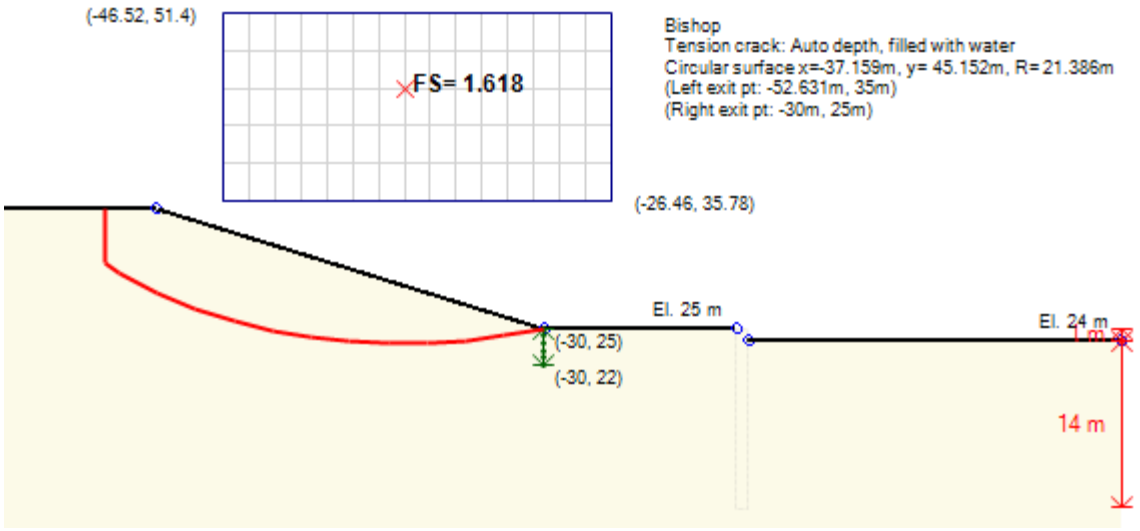


Figure 4.4.1: Viewing the critical slope stability rupture surface

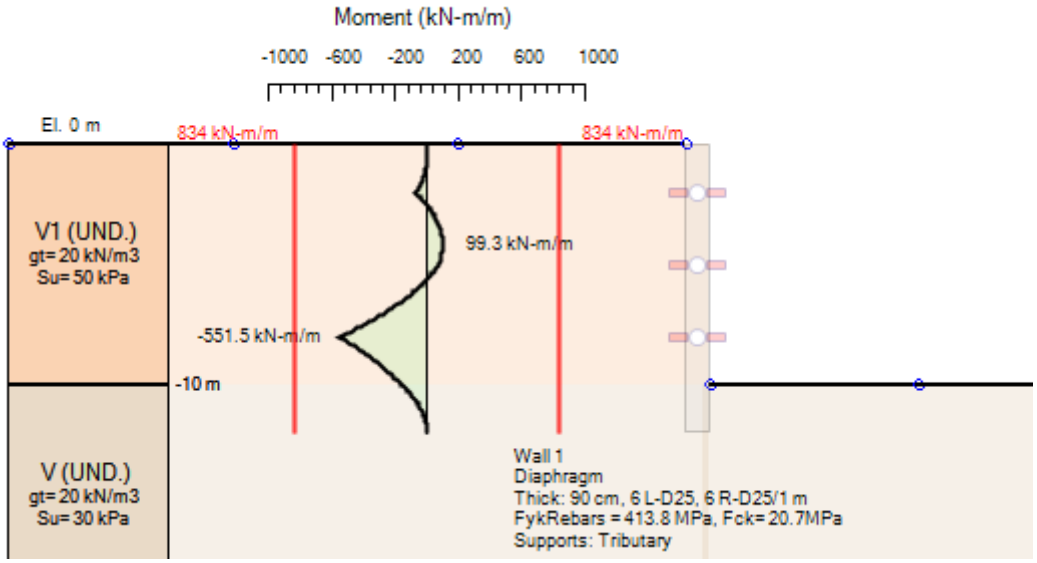


Figure 4.4.2: Typical moment diagram.

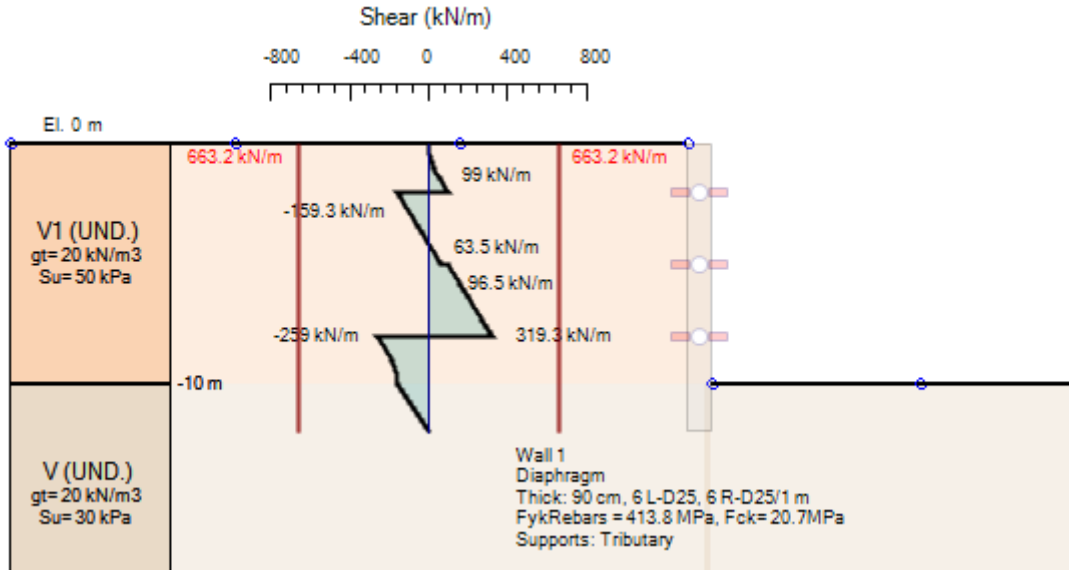


Figure 4.4.3: Typical shear diagram.

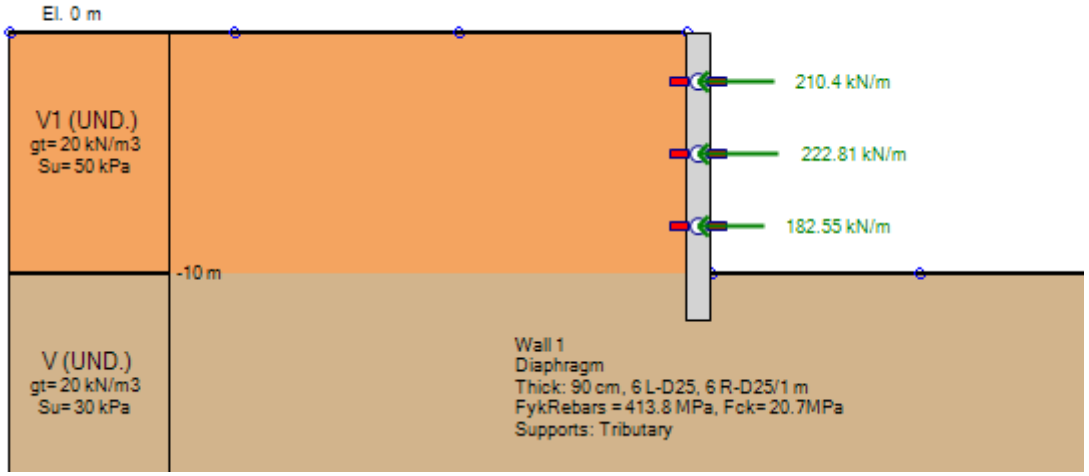


Figure 4.4.4: Support reactions.

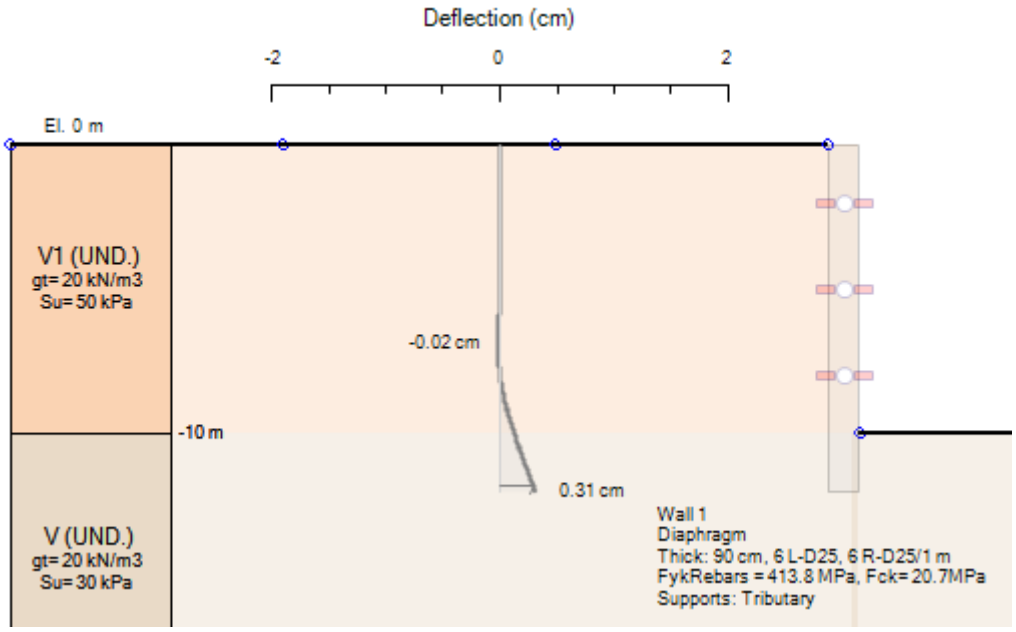


Figure 4.4.5: Wall deflection.

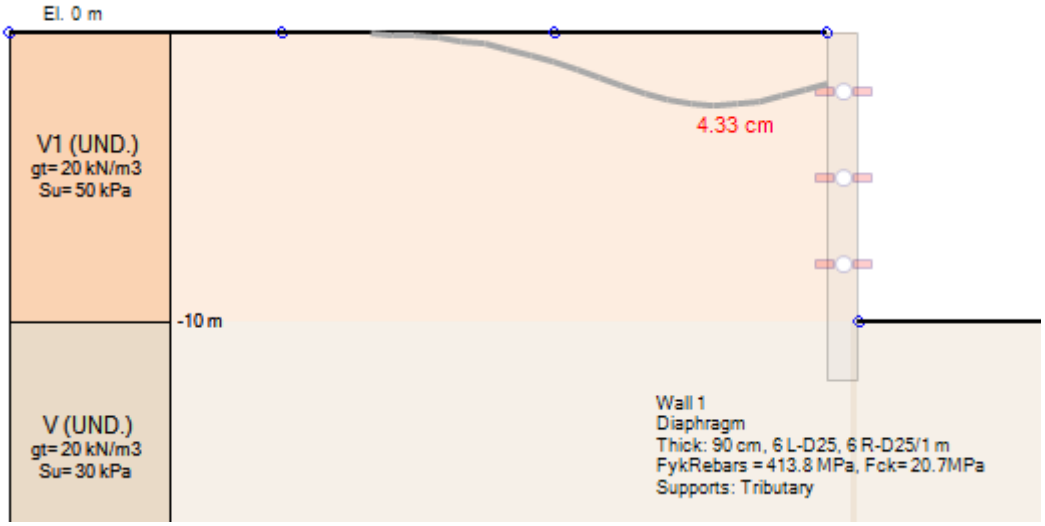


Figure 4.4.6: Surface settlement.

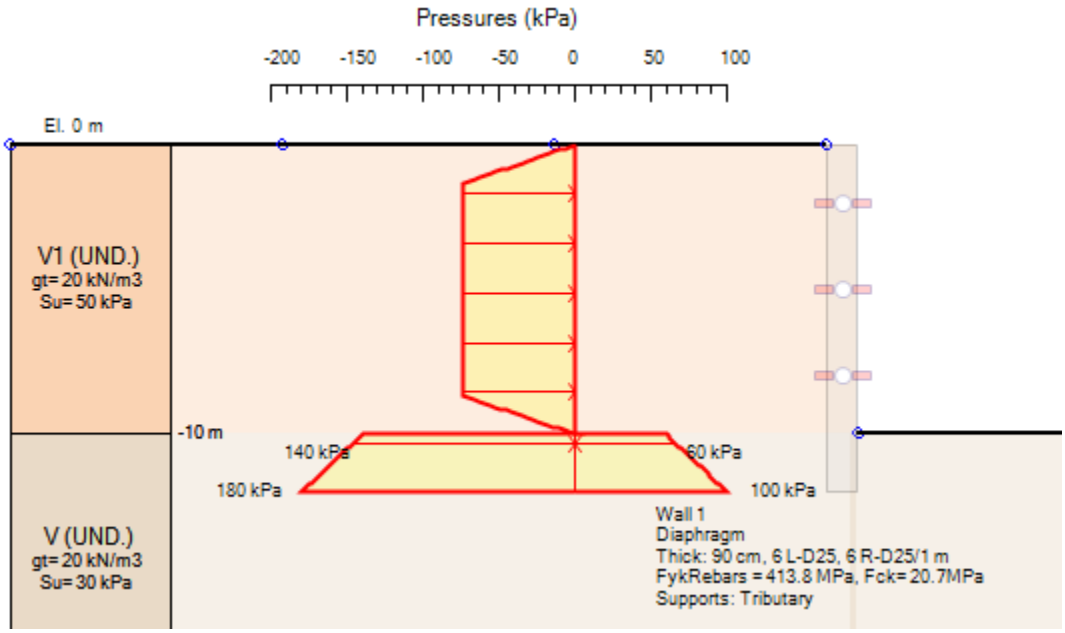


Figure 4.4.7: Effective horizontal soil pressures (limit equilibrium).

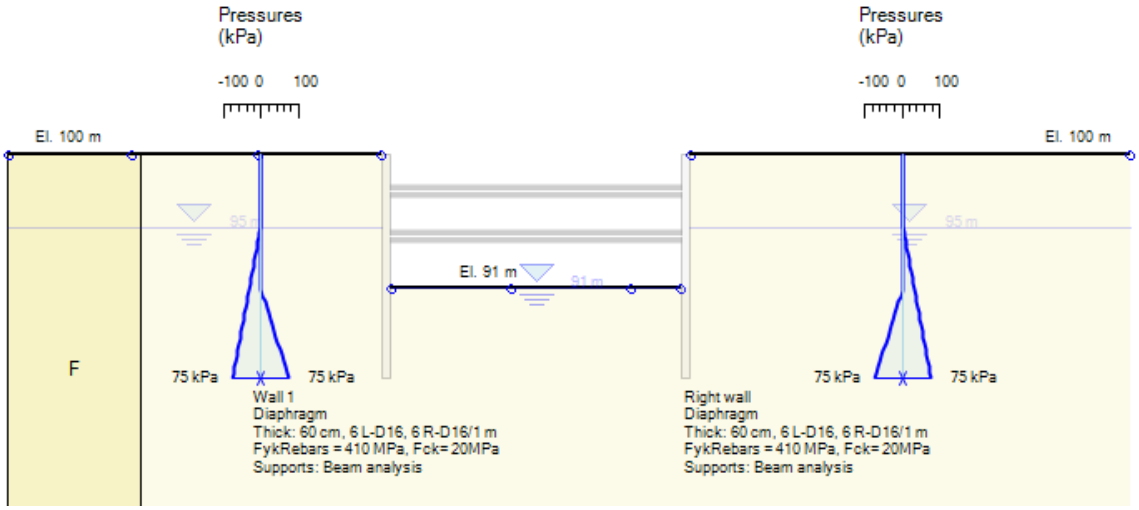


Figure 4.4.8: Water pressures.

4.5 Report Options (Printed Reports)

Once a project is analyzed, full analysis reports can be generated by selecting the Reports – Options option at the Report tab. By selecting this, we can modify the included output sections. On the left side of the dialog, we can select which design sections and stages will be included in the current report (Figure 4.5.1). From the Available Report Sections area, we can select the results and options that shall be included (Figure 4.5.2). We can also drag and drop these items at the Report Format area (Figure 4.5.3). Next, the user can select to see a preview of the report and export it in a word or PDF format at the area indicate in Figure 4.5.3.

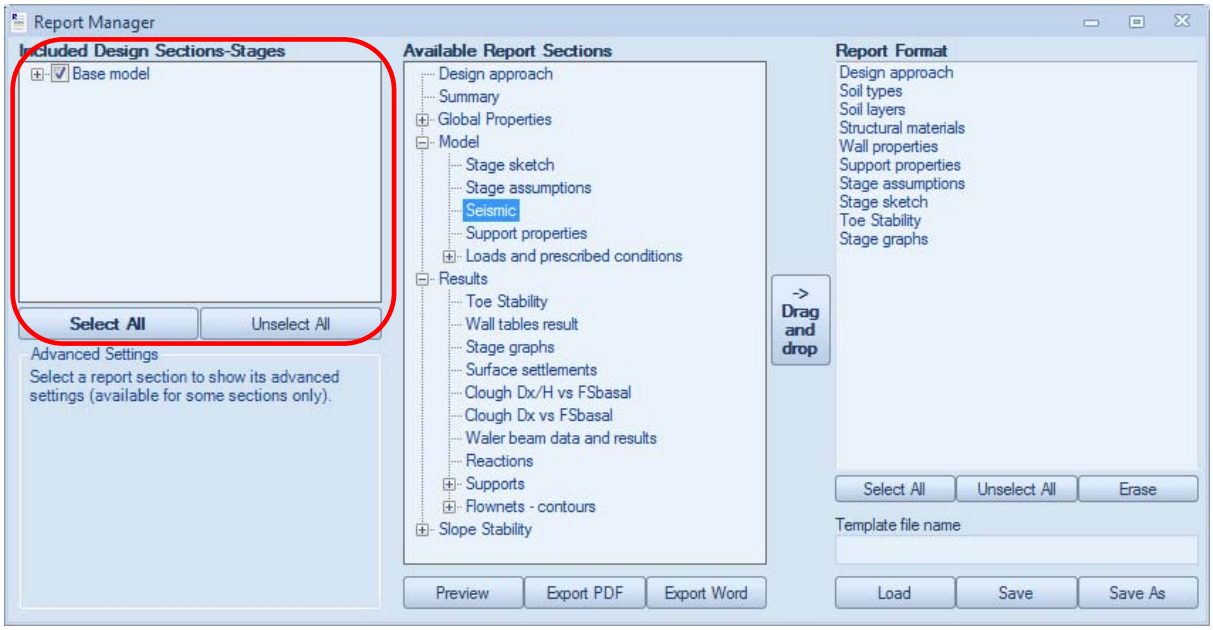


Figure 4.5.1: The included Design Sections area.

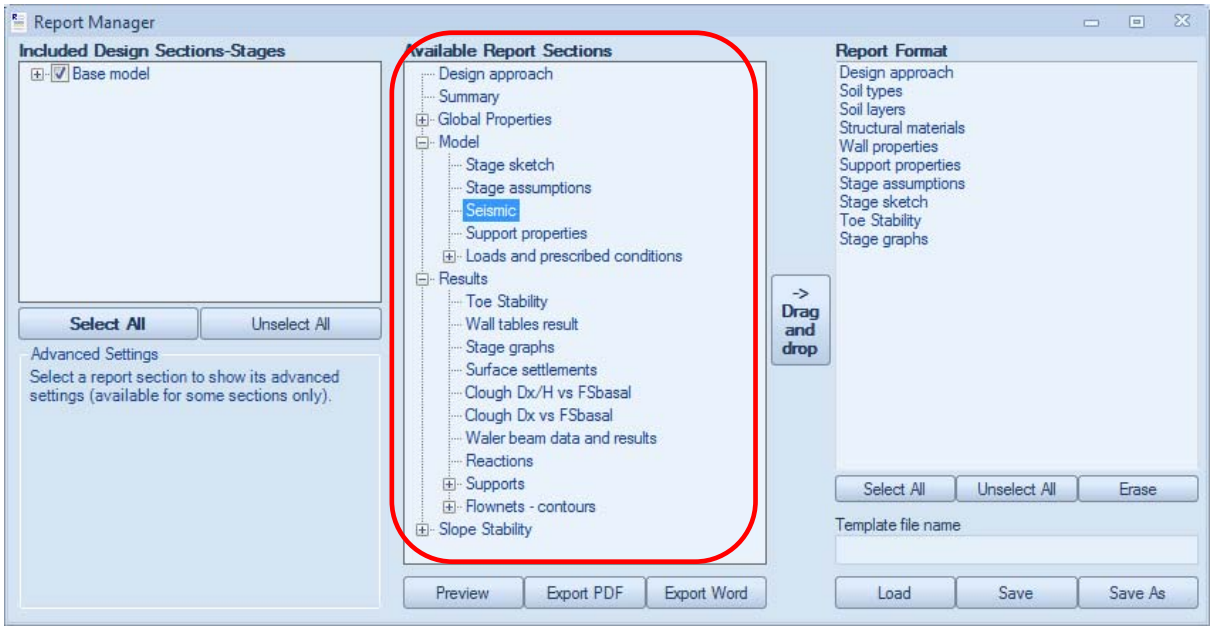


Figure 4.5.2: The Available Report Sections area.

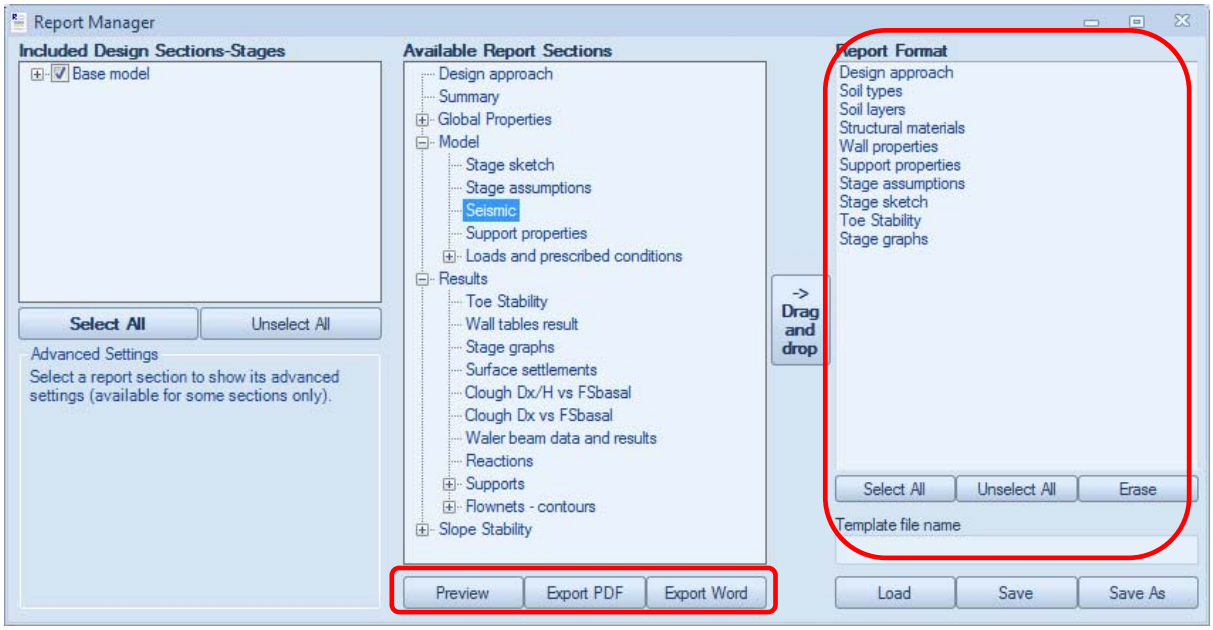


Figure 4.5.3: The Report Format area and the preview and export buttons.

4.6 Footing Loads (3-Dimensional Loads)

In many cases the effects of three dimensional loads on the wall have to be evaluated. In DeepXcav footings can be used to define three dimensional loads that can vary in magnitude for any desired stage. DeepXcav offers a number of useful alternatives for evaluating the effect of three-dimensional loading on excavation walls. The theoretical solutions of elasticity are truly applicable for level surfaces, however, in the absence of any other exact closed form solution many practicing engineers use these methods for non-level ground surfaces. Caution should be exercised and the limitations recognized when dealing with non-level surfaces.

Parameter	Description
X	The x coordinate of the footing (center)
Y	The y coordinate of the footing (out-of-plane) or 3D coordinate at the footing center.
Z	Elevation (the elevation is automatically adjusted for surface footings) at the current surface elevation for each stage.
T	Footing thickness (not used for calculations)
Bx	Footing width along X axis.
By	Footing width along Y axis (out-of-plane, or into screen)
Θ	Counter clockwise in-plan footing rotation in degrees.
L	Column length (graphical)
Treat footing as point load	Select this option to model the footing as a point load for calculating lateral surcharges. With this option the Boussinesq solution is used when the footing is at the surface, and Mindlin's solution is used when the footing is below the surface. A Poisson's ratio of 0.25 is used. When this option is not selected, then DeepXcav uses the Holl 1940 solution for rectangular loads as outlined in Poulos & Davis, 1974. This solution is independent of the Poisson's ratio. The method is applicable for loads that are on a level surface, however, in the absence of a detailed closed form solution for rectangular loads beneath the ground surface Holl's solution is also used for loads beneath the ground surface.
Average load over support	Select this option if you want to average the load over the maximum support spacing. With this option DEEP averages the surcharge from (y - maximum support spacing/2) to (y+maximum support spacing/2). The

spacing	surcharge is calculated every Dyw increment. A minimum of nMin intervals are used.
DX	(Not used in this version)
DY	(Not used in this version)
nMin	Minimum number of intervals along y axis for computing stresses.
Dyw	Interval for calculating stresses when load is averaged over maximum support length.

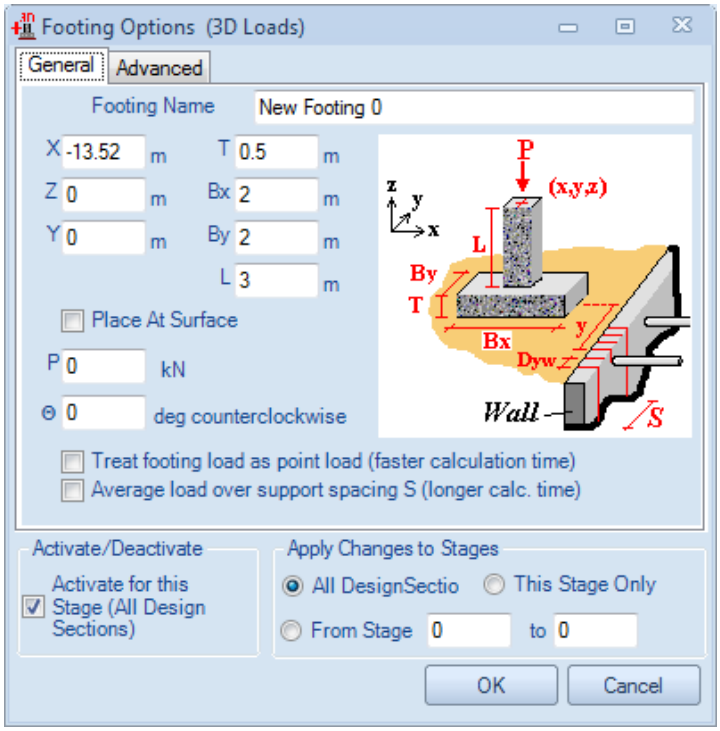


Figure 4.6.1: The footing options dialog General tab.

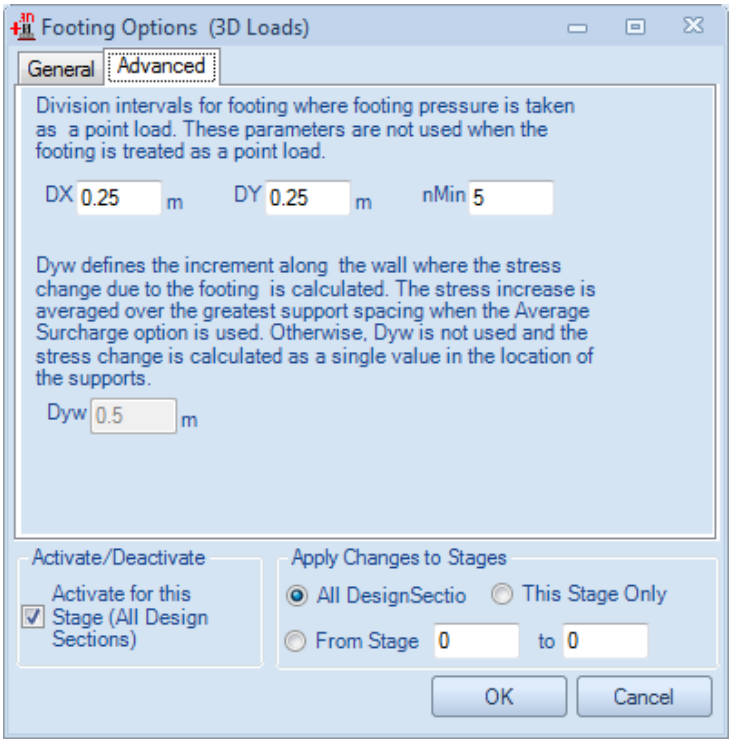

Footing Options (3D Loads)
General
Advanced
Division intervals for footing where footing pressure is taken as a point load. These parameters are not used when the footing is treated as a point load.
DX 0.25 m
DY 0.25 m
nMin 5
Dyw defines the increment along the wall where the stress change due to the footing is calculated. The stress increase is averaged over the greatest support spacing when the Average Surcharge option is used. Otherwise, Dyw is not used and the stress change is calculated as a single value in the location of the supports.
Dyw 0.5 m
Activate/Deactivate
Activate for this Stage (All Design Sections)
Apply Changes to Stages
All DesignSectio
This Stage Only
From Stage 0 to 0
OK
Cancel

Figure 4.6.2: The footing options dialog Advanced 0ptions.

4.7 Buildings & Building Wizard

Excavations are often constructed next to existing buildings that can potentially exert significant loads on the retaining system. Thus building loads have to be properly accounted for. Estimating building loads can be a very time consuming effort. DeepXcav can make such estimations much easier with the building wizard option.

To create a new building simply select the icon from the left tool bar and click on the desired point where you want to add the building. Once you do so the building wizard launches as shown below:

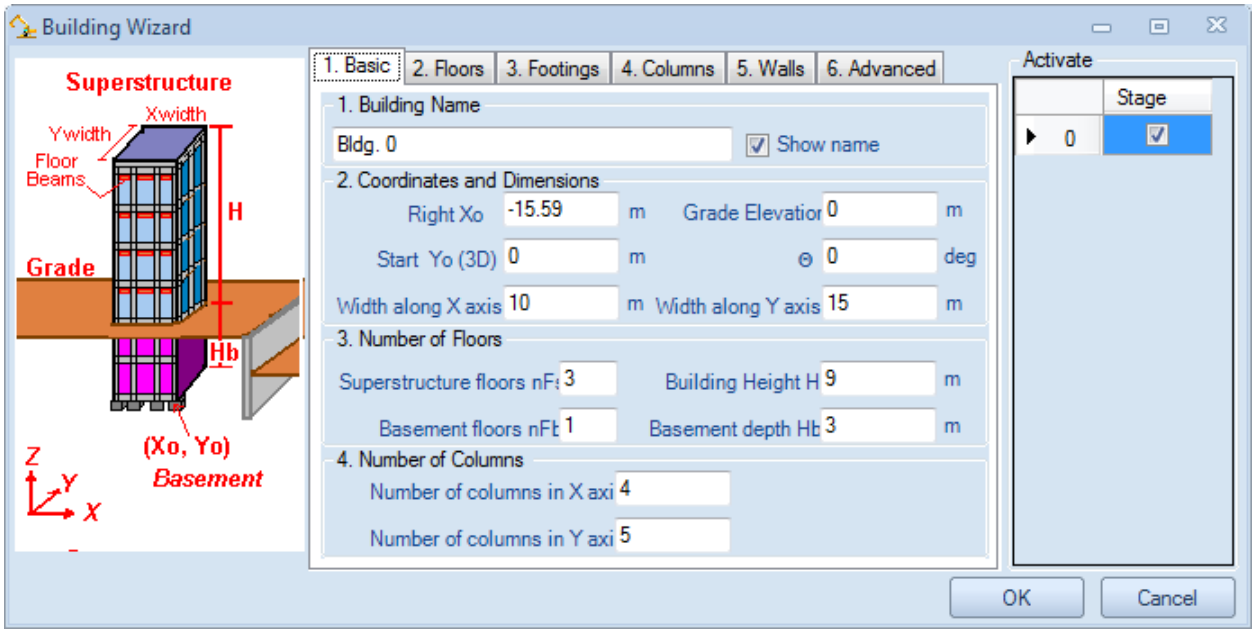


Figure 4.7.1: General building dimensions.

Parameter/Icon	Description
	Adds a three dimensional building
Xo	The right x coordinate of the building (center)
Yo	The y coordinate of the footing (out-of-plane) or 3D coordinate at the footing center.
Grade Elevation	Elevation of the ground floor.
Θ	Building rotation.
Width along X axis	Building width along the x axis direction.
Width along Y axis	Building width along the y axis direction (out of plane)
nF	Number of superstructure floors including the ground floors not including the building roof.

nFb	Number of basement floors (below grade elevation)
Building height H	Building height above grade elevation
Basement depth Hb	Basement depth to top of footings (bottom of base slab)
Number of columns in X	Number of building columns in X direction
Number of columns in Y	Number of building columns in Y direction

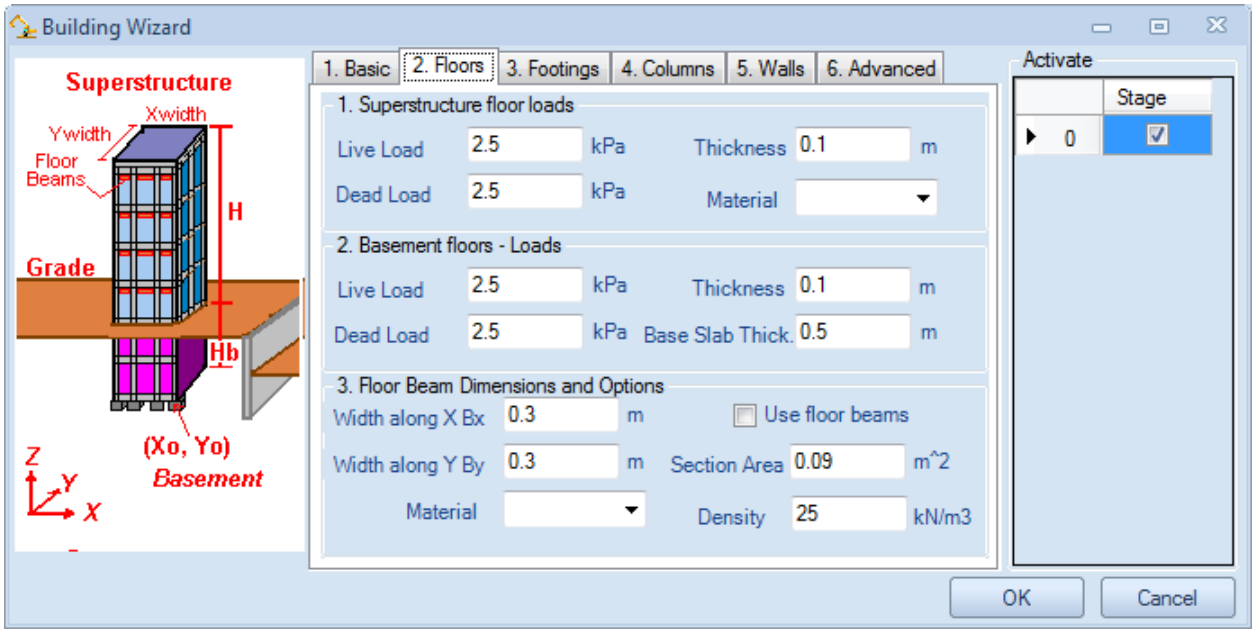


Figure 4.7.2: Floor loads and options.

Parameter/Icon	Description
Floor Live load	Live load on each floor (may be different for basement floors)
Floor Dead load	DEEP does not automatically calculate the dead load of each floor based on the thickness. While such a calculation is easy it is felt that a proper estimation of the floor dead load has to be manually performed and must include many items such a tiling etc. that are outside the scope of the current version of DEEP. A different dead load can be included for the basement floors.
Floor Thickness	Floor thickness for graphical reasons only
Use floor beams	Check this option to use floor beams under each side of each floor between columns. The dead load of each floor beam is calculated based on its density and section area.
Floor beam Bx	Building width along the x axis direction.
Floor beam By	Building width along the y axis direction (out of plane)
Floor beam Section area	The section area can be manually changed to include hollow beams and non rectangular shapes. This area is used for dead load calculations.
Floor beam density.	The floor beam density is initially set equal to the concrete density. You may use a different density to simulate other materials.

All loads in DeepXcav are eventually transferred to rectangular footings. While not every footing in the real world is rectangular, this simplification captures most cases. The 3rd tab gives the option to define the footing dimensions (Figure 4.8.3). Please note that DeepXcav assumes that all footings are made of concrete and calculates the dead load of each footing based on its dimension. Also, DeepXcav gives the option to include grade beams that connect all building footings.

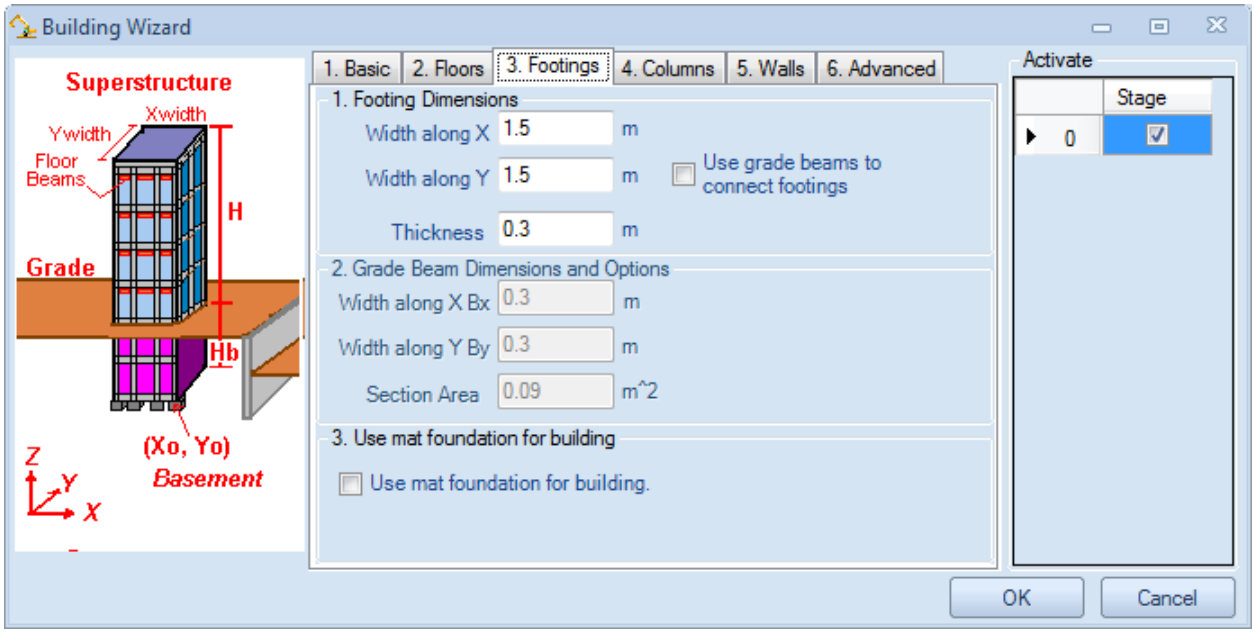


Figure 4.7.3: Footing options for buildings.

Parameter/Icon	Description
X	Width along X axis footing dimension
Y	Width along Y axis footing dimension
Thickness	Footing thickness
Use grade beams	Check this option to use grade beams to connect footings
Grade beam Bx	Grade beam width along the x axis direction.
Grade beam By	Grade beam width along the y axis direction (out of plane)
Use mat foundation for building	Check this option to use a mat foundation for this building

Floor loads are eventually transferred to footings with the help of building columns. Column dimensions and other parameters can be defined from the 4th tab (Figure 4.8.4).

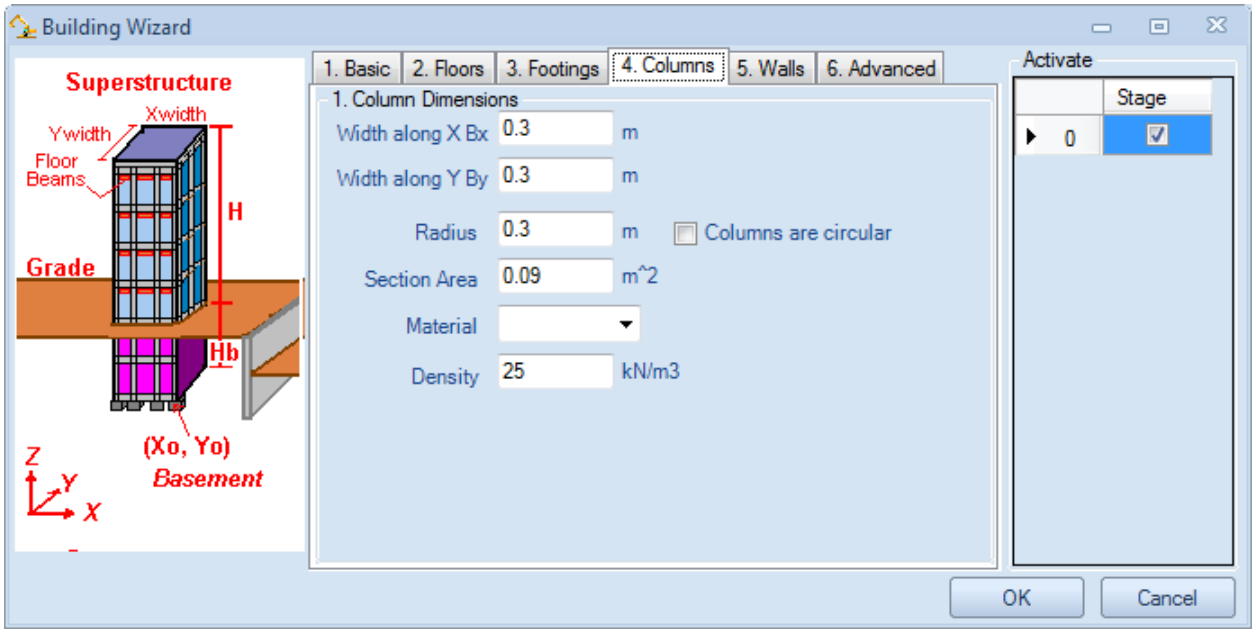


Figure 4.7.4: Building column options.

Parameter/Icon	Description
Bx	Width along X axis column dimension
By	Width along Y axis column dimension
Radius	Column radius (if the circular columns option is selected)
Section Area	The section area of the column
Material	The material of the column
Density	The density of the column

Most buildings have walls. In order to better simulate building loads, DeepXcav gives you the option to simulate different exterior, interior and basement walls. Wall loads are calculated from the wall area times the density. The open wall space can be used to simulate doors, windows, and other openings that do not have any dead load.

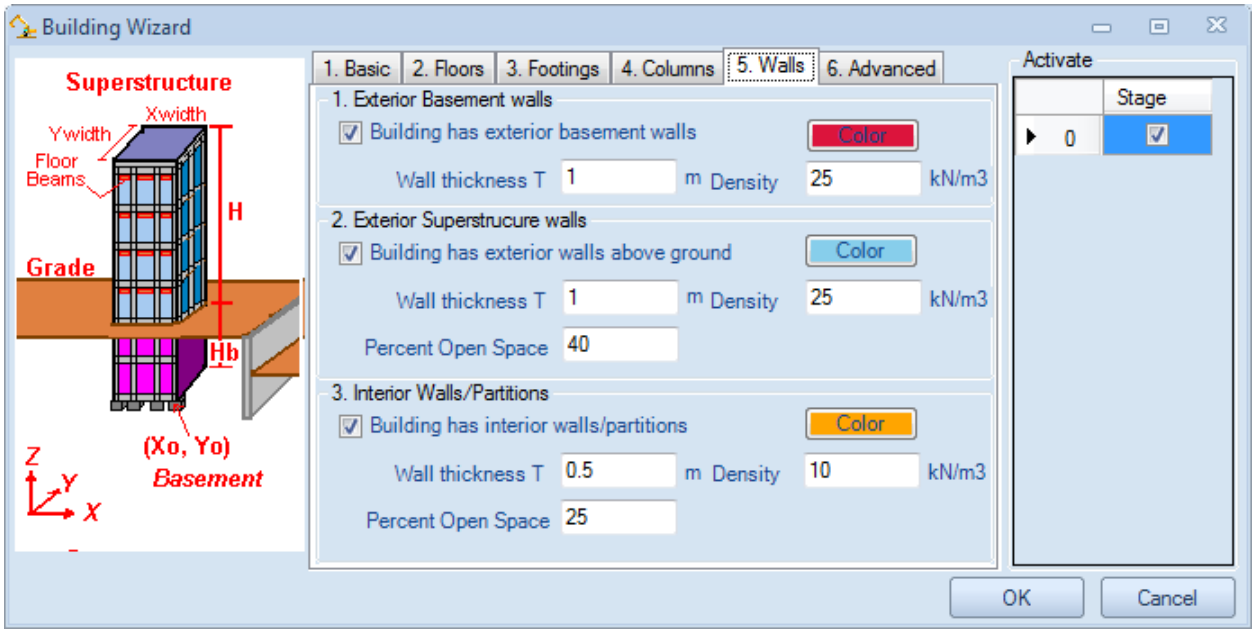


Figure 4.7.5: Building wall options.

Here the user has the following options:

- Building has exterior basement walls. If the user checks this option, he can define the walls thickness and density, as well as the wall's color.
- Building has exterior superstructure walls. If the user checks this option, he can define the walls thickness and density, as well as the wall's color.
- Building has interior walls and partitions. If the user checks this option, he can define the walls thickness and density, as well as the wall's color.

In the advanced tab (Figure 4.8.6 below), we have the option to include relief load calculations. By checking this option, DeepXcav removes the effect of soil's dead load on the surcharge stresses since this soil is removed in order to create the basement. This is done with elastic solutions by averaging the vertical stresses on all building corners.

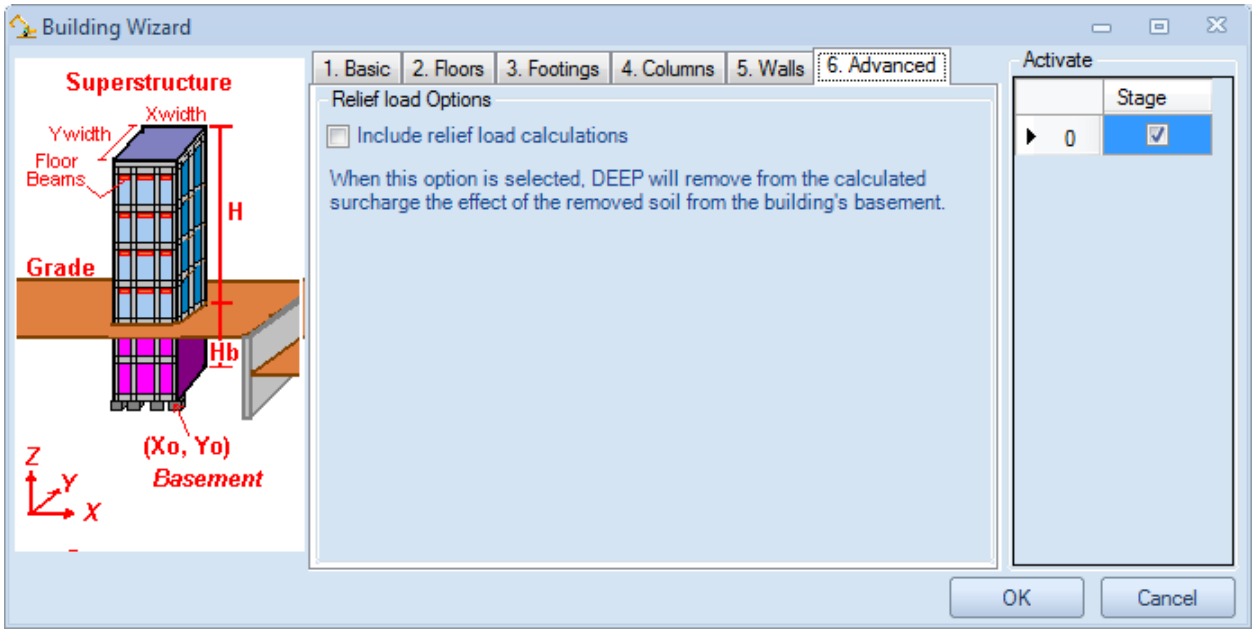


Figure 4.7.6: Advanced building options.

Building footings can be edited by right clicking on them from the plan view (Figure 4.8.7). If a footing is moved or its dimensions changed, and building loads will be changed (if the user decides to do so). All column loads are calculated based on approximate tributary areas from each supported floor. **At the same time, the building will be saved is a long format since it is not fully rectangular.**